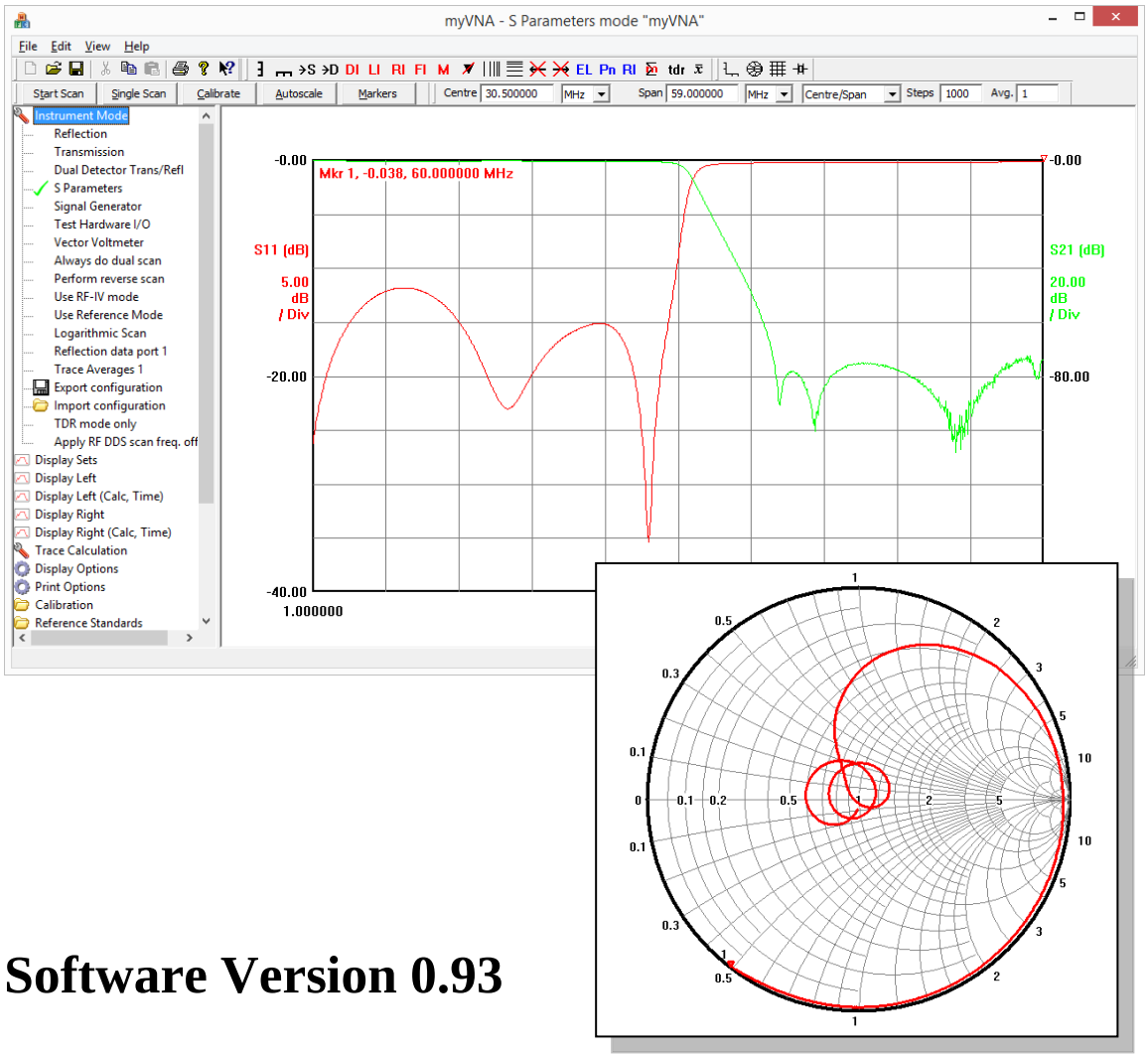
myVNA
User manual



Software Version 0.93

User Manual version 1.05, 5th July 2015

Copyright © Dave Roberts G8KBB 2015

1 Introduction

This document explains how to use myVNA. It gives step by step instructions in the form of a short tutorial from initial installation, showing how to use each of the main features of the program.

For an overview of how to install the program, please refer to Section 3, “Start Here” on page 7.

myVNA is a Windows GUI for the N2PK VNA.

myVNA offers the following:

- Reflection, Transmission or Dual detector scans
- S Parameter measurements with 5,6,10 or 12 term correction
- Automated Crystal Equivalent Circuit measurement
- Full transverter support
- Ability to load & store traces in multiple file formats
- Full set of display modes including Smith Charts and Polar displays
- Signal and sweep generator mode
- Integrated support for parallel and USB connection
- Runs under W98, ME, W2000, XP, Vista, Windows 7 & 8
- Basic, CDS, Harmonic Suppression and 2 / 3 / 4 / 5 harmonic modes
- Manual and automatic tracking markers with marker arithmetic
- Scans of up to 50,000 points per scan
- Averaging of trace sequences or averaging up to 10,000 readings per point
- Print & Windows clipboard copy support
- Switch and attenuator functions with manual or automated switching for S parameter test set.
- Test mode for I/O lines in parallel or USB mode
- RF IV & Reference Modes
- Logarithmic or linear scans and displays
- Drag & drop file import
- Network Client / Server for remote operation
- Vector Voltmeter (VVM)
- On screen cursors may be displayed in rectangular displays
- Arbitrary equation editor allows user defined functions to be created
- Basic network simulation
- TDR mode
- An Automation (OLE) interface
- A “custom” hardware interface for control of other VNA hardware

2 Acknowledgements

There are many people who deserve to be acknowledged for their contribution. The following are noted:

First, above all, to Paul Kiciak, N2PK, for producing a most excellent VNA design and for help, encouragement, algorithms and advice. Paul together with Wayne Torrey also own the design for the digital filter used for the harmonic mode filtering

Ivan Makarov VE3IVM similarly needs to be thanked for his suggestions, testing, signing of drivers, producing good PCBs and the RF-IV hardware modules

To Jack R. Smith K8ZOA for the paper “Crystal Motional Parameters A Comparison of Measurement Approaches”, the test crystals and the transmission jig.

My thanks to Mike Collins for the suggestions on optimisation of the calibration flows and to Roderick Wall VK3YC my thanks for the feature requests, testing, feedback and the worked examples of filtering, matching and tuning using TDR mode.

The program “about” box also acknowledges contributions from others in regard of the code – a copy is shown below.

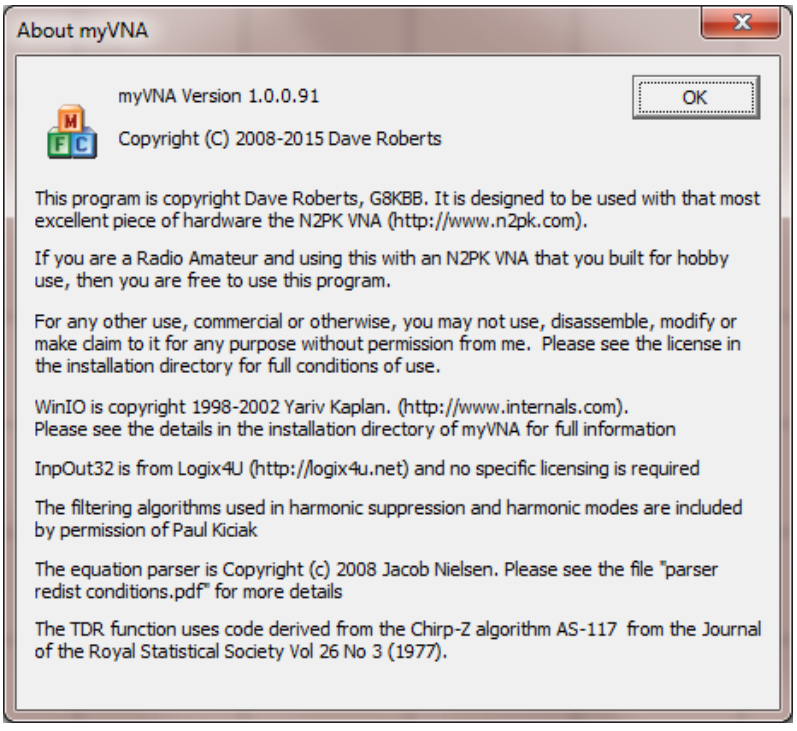

About myVNA
myVNA Version 1.0.0.91
OK
Copyright (C) 2008-2015 Dave Roberts
This program is copyright Dave Roberts, G8KBB. It is designed to be used with that most excellent piece of hardware the N2PK VNA (http://www.n2pk.com).
If you are a Radio Amateur and using this with an N2PK VNA that you built for hobby use, then you are free to use this program.
For any other use, commercial or otherwise, you may not use, disassemble, modify or make claim to it for any purpose without permission from me. Please see the license in the installation directory for full conditions of use.
WinIO is copyright 1998-2002 Yariv Kaplan. (http://www.internals.com).
Please see the details in the installation directory of myVNA for full information
InpOut32 is from Logix4U (http://logix4u.net) and no specific licensing is required
The filtering algorithms used in harmonic suppression and harmonic modes are included by permission of Paul Kiciak
The equation parser is Copyright (c) 2008 Jacob Nielsen. Please see the file "parser redist conditions.pdf" for more details
The TDR function uses code derived from the Chirp-Z algorithm AS-117 from the Journal of the Royal Statistical Society Vol 26 No 3 (1977).

Table of Contents

1 Introduction....2
2 Acknowledgements....3
3 Start Here....7
3.1 Installation....7
3.2 Product Keys....8
3.3 Product Activation....8
4 Tutorial....10
4.1 First Run....10
4.2 Setting Initial Hardware Settings....11
4.2.1 N2PK Hardware Setup....12
4.2.2 MSA Hardware Setup....13
4.3 Hardware Interface Selection....14
4.4 Choose CDS / Harmonic mode....16
4.5 Setting the ADC mode....17
4.6 Setting the Display Options....17
4.7 Performing your first scan....17
4.7.1 Calibration....19
4.7.2 Scanning....22
4.7.3 Dragging the display around....23
4.7.4 Markers and Marker Arithmetic....26
4.8 Data Stores....30
5 Managing the display....31
5.1 Popup Menus & Toolbars....31
5.2 Displaying Multiple Traces....34
5.3 Scan setting options & the Scan Toolbar....35
5.4 Labelling the Screen....35
5.5 Dragging Items on the Screen....37
5.6 Locking the Scan and the Display....38
5.7 On Screen Cursors....38
5.8 Markers and Marker Arithmetic....42
5.8.1 Global Show / Hide Markers....46
5.9 Full Screen Display....46
5.10 Keyboard Shortcuts....46
6 Save, print and copy....47
6.1 Saving Data to and loading data from Disk....47
6.1.1 Autosave....49
6.1.2 Load and Store options and configuration....49
6.1.3 Export & Import Program Configuration....49
6.2 Printing....50
6.3 Copy to Clipboard....51
6.4 Drag & Drop....52
7 Program Limits and other options....53
8 Smith and Polar Charts....56
8.1 Zooming....58
9 Report View....60
10 Equivalent Circuits....62
10.1 Crystal Motional Parameters....63
11 Dual Detector Scan....68

12 Logarithmic & Linear Scans 71
13 S Parameters 73
14 Harmonic modes and CDS 77
15 Group Delay 79
16 Transverters 80
17 Attenuator and Switch control 81
17.1 Toggling switches during a scan 82
18 Open/Short/Load Calibration Standards 83
18.1 Loading & Saving Calibration Standards Data 83
18.2 Saving Calibration Standards in Calibration data files 83
18.3 Saving System Reference Impedance in Calibration & Standards Files 83
18.4 LLC (OSLC) Calibration 84
18.5 Calibration Notes 84
18.6 Loading / saving settings in calibration data 84
19 Unguided Calibration 85
20 Reference Planes 87
21 RF IV Method 88
22 Reference Mode 89
23 TDR mode 90
23.1 Using Time Domain to tune variable coupling BPFs 93
24 Marker Measurements 97
25 Trace Averaging 100
26 Signal Generator 101
27 Vector Voltmeter 103
27.1 DDS Reset Test 106
28 User Defined Display Functions 108
28.1 Data Sources 109
28.2 Functions 112
28.3 Numbers 113
28.4 Values 113
29 Network Simulation 114
29.1 Load & Save Simulations 120
29.2 Impedances 120
29.3 Filter matching example 120
29.4 Simulation and Crystal Equivalent Circuits 123
29.5 Simulation Limitations 124
30 Multiple Program Configurations 125
31 Multiple USB connected VNAs 126
32 OLE Automation 127
33 Network Client / Server 128
34 DDS Power Down 130
35 Command Reference 132
35.1 Main Menu 132
35.1.1 Main Menu - File 132
35.1.2 Main Menu – Edit 133
35.1.3 Main menu – View 133
35.1.4 Main menu – Help 134
35.2 Toolbars 134
35.2.1 Windows Toolbar 134
35.2.2 Controls Toolbar 134

35.2.3 Display Format Toolbar 136
35.2.4 Scan Control Toolbar 136
35.2.5 Span Toolbar 136
35.2.6 MSA Path toolbar 137
35.3 Treeview controls 137
35.3.1 Instrument Mode Treeview Node 138
35.3.2 Display Treeview Nodes 139
35.3.3 Trace Calculation Treeview Node 141
35.3.4 Display Options Treeview Node 142
35.3.5 Print Options Treeview Node 142
35.3.6 Calibration Treeview Node 143
35.3.7 Reference Standards Treeview Node 144
35.3.8 Load & Save Treeview Node 145
35.3.9 VNA Hardware Treeview Node 146
35.3.10 Choose Trace Colours Treeview Node 148
35.3.11 Transverters Treeview Node 148
35.3.12 Network Settings Treeview Node 148
35.3.13 Switches and Attenuators Treeview Node 148
35.3.14 Debug Treeview Node 149
35.4 Popup Menus 150
Annex A - Common Configurations 152
Annex B - S Parameter Test Sets and Bridges 155
Annex C - Glitches when using the parallel port 156
Annex D - Automation interface example contents 158
Annex E - Custom VNA Interface 169
Annex F - How fast will it scan? 171
USB Timings 171
Parallel Port 173
Annex G - Configuring the USB hardware 175
Annex H - Testing I/O Lines 178
Index 179

3 Start Here

This chapter describes how to install the program including entry of product keys and program activation.

3.1 Installation

First, install the program from the setup.exe that accompanied myVNA.msi. If you have a previously installed version, running the new installer will remove the old version automatically first. It will also remove all settings from the old version. The installer will create shortcuts in the start menu and desktop. It should also create a directory in "my documents" called myVNA; it is suggested that you store settings there. The program may be uninstalled either by rerunning the installer or from add/delete programs in Control Panel. It also copies into the program directory some images (used in calibration and equivalent circuits) and some DLLs. One is inpout32.dll for parallel port access. Another is vnadll.dll, the USB interface dll. In order to run the USB interface you must also have installed the USB drivers. To support the MSA there is also a specific dll – msadll.dll. Copies of the current versions of each are included in the installation and should appear as zip files in the installation directory.

The installer also creates a subfolder with USB drivers and a utility to program the USB controller chip. When the VNA is connected, if USB is used, Windows will prompt for the appropriate drivers. DO NOT let windows search online for drivers, make sure you tell windows to let you choose the drivers from local storage, and navigate to the appropriate subfolder in the USB drivers folder. Installation of the drivers is a two-step process – it will ask you twice, point it to the same location both times. For more detail, please refer to Annex G - or to the USB driver installation instructions for which there should be a shortcut in the start menu.

For 64 bit operating systems, Vista or Windows 7 & 8, signed parallel port drivers are needed if you wish to use the parallel port to drive the VNA. In this case, you must install the drivers first by executing the file "InstallDriver.exe" which you will find in the program installation folder. It needs to run as administrator but should only need to be run once. To run this as administrator, right click on the file and select "run as administrator".

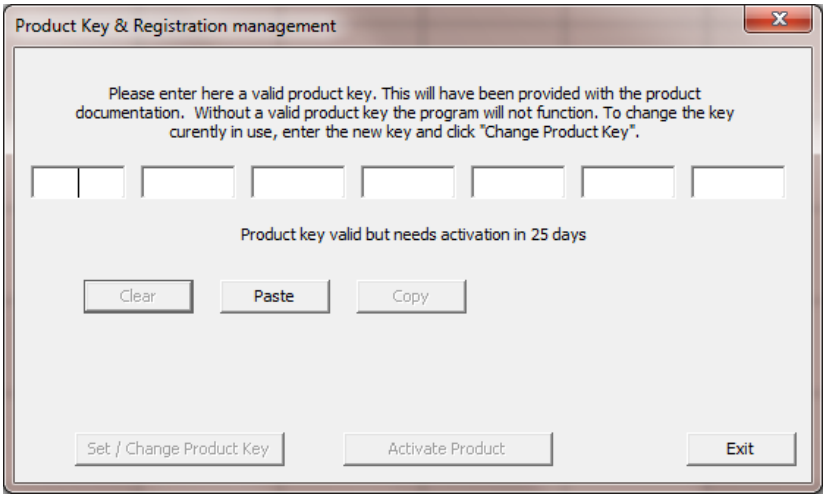
There is a choice of parallel drivers. Support is built in for both inpout32 and WinIO. The former appears more compatible, the latter faster. If WinIO works, use it, otherwise try inpout32. WinIO will only work under Windows Vista if you have local admin rights to the PC.

For the N2PK, both USB and parallel are supported, for the MSA only USB is available.

The program also supports a "Custom" VNA interface. This allows an interface to be written to other VNA hardware so that it can be controlled by myVNA. This is described in Annex E.

3.2 Product Keys

The first time you run the program it will ask for a product key. If you use the program for personal Amateur Radio purposes, and not for commercial purposes, then product keys are available free from myvna@btinternet.com and for commercial use the product key will have come with the product information.


Product Key & Registration management
Please enter here a valid product key. This will have been provided with the product documentation. Without a valid product key the program will not function. To change the key curently in use, enter the new key and click "Change Product Key".
Product key valid but needs activation in 25 days
Clear
Paste
Copy
Set / Change Product Key
Activate Product
Exit

To paste in a product key, which will look something like this:

IZY9G-HUBIB-4RUY6-9REMN-34IHP-NE3PY-B9BNQ

use the "Paste" button, having previously copied the product key into the clipboard. Alternatively, the key may be entered manually.

The product key comprises letters A-Z and numbers 2-9. The numbers 0 and 1, and the letters L and O are not used.

The program will indicate simple typographical errors if they are detected, but the key will not be cryptographically validated until you set the key and exit from the dialog. When an apparently valid key is entered, the "Set/Change Product Key" button will be enabled. Click it then click on "Exit". The next time you open the dialog box it will indicate the status of the product key (where it says "The product has been successfully activated already" in the above example).

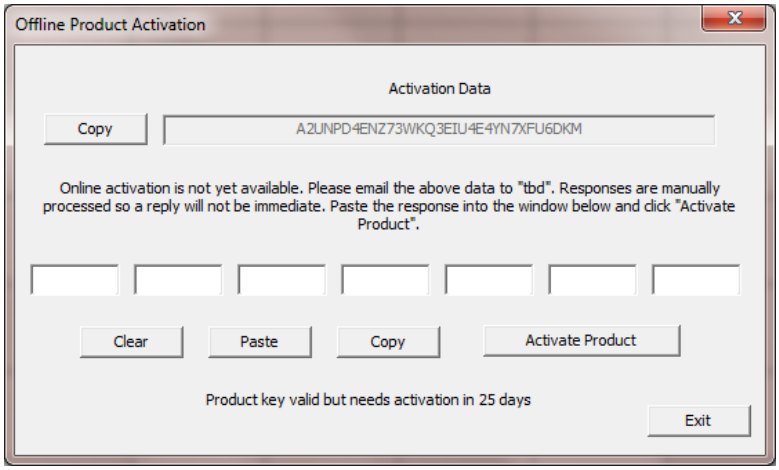
Depending on the product key it may also be necessary to Activate the program. If so, then the number of days until it must be activated will be shown in the dialog as above.

At the moment, automatic activation is not available – it must be done by email so don't leave it until the last minute.

3.3 Product Activation

If needed you will normally have up to 30 days to activate the product (60 days for commercial licenses). Automatic (online) activation is not currently available so manual activation must be used.

From the “Help” menu, there are options to enter or change the product key or activate the product. The “Activate” option opens a dialog as follows:


Offline Product Activation
Activation Data
Copy
A2UNPD4ENZ73WKQ3EIU4E4YN7XFU6DKM
Online activation is not yet available. Please email the above data to "tbd". Responses are manually processed so a reply will not be immediate. Paste the response into the window below and click "Activate Product".
Clear
Paste
Copy
Activate Product
Product key valid but needs activation in 25 days
Exit

If a valid product key has been entered that needs activation, then activation data will be displayed. Copy this into an email to the address shown (or to myvna@btinternet.com if one is not otherwise indicated). Upon receipt of the activation key, paste it in and click “Activate Product”.

Activation keys look like product keys but they are not interchangeable. Do not try to enter a product key as an activation key or vice versa.

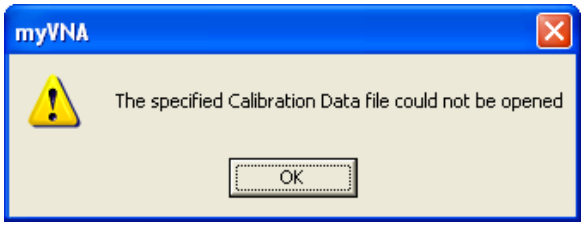
Please note that given the security sometimes applied to computer settings for some operating systems, it may be the case that if activated under one user account on a shared computer it may not appear to be activated for other users of the same computer. If this is the case, use the same product and activation key for other accounts. It may also be the case that due to local security policy restrictions, you need to run myVNA as administrator when performing product key entry or activation.

4 Tutorial

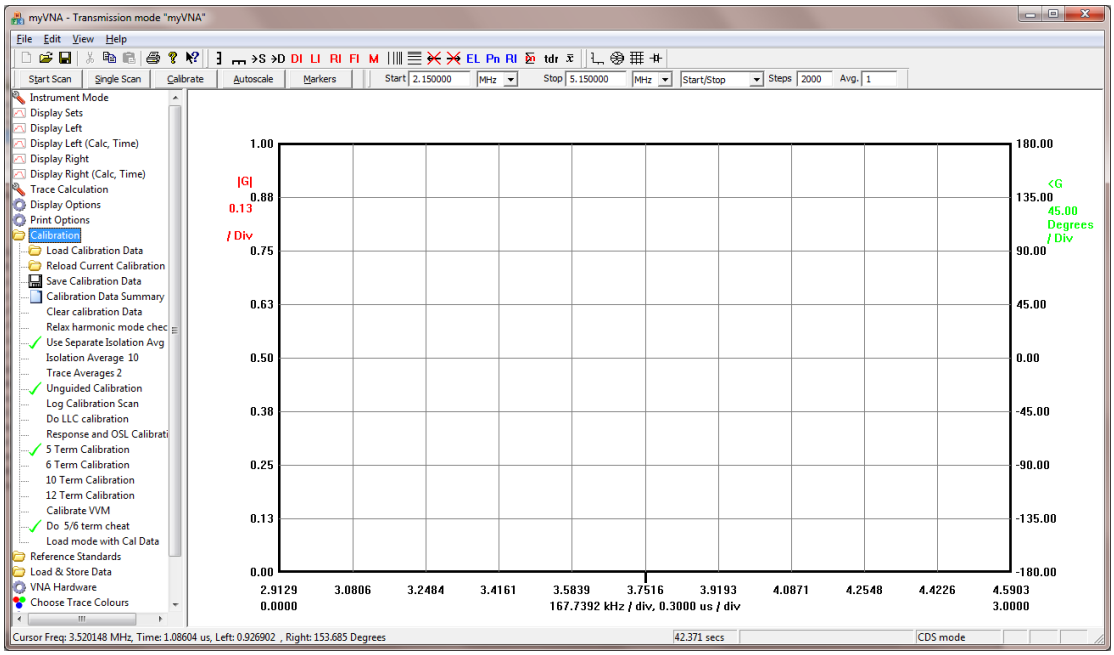
This chapter provides a step by step tutorial on using myVNA.

4.1 First Run

When you first run the program, or after installing a new version, it will not find any stored parameters in the registry (it does *NOT* use ini files). Everything will default and, specifically, it will not find any calibration data. If this is the first time you have run the program, or have run it but never performed a calibration scan it will silently proceed to the main screen, otherwise it will tell you this:



If this is shown, just click OK and proceed to the main menu as shown below:



The screen is divided into several parts. At the top are 5 toolbars – the usual windows one, a scan control toolbar with buttons on it, a controls toolbar, a display format toolbar and a span toolbar.

All are floating, dockable toolbars and can be moved around. When you exit the program, the position will be remembered. It won't do a perfect job of this. It won't let you put them all on one row for example, because this interferes with the menu command to hide the main toolbar. You can't hide the scan and span toolbars!

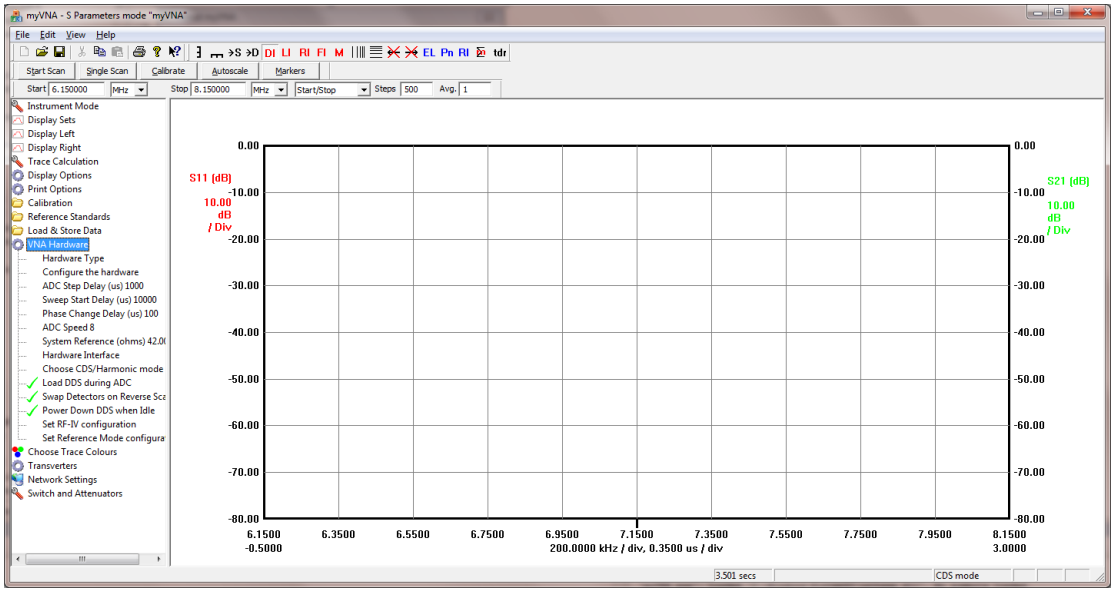
On the left is a "treeview" of the main settings. On the right is the output display. Initially, if you position the mouse over one of the entries in the treeview on the left, it

will display an “Info Tip” with a brief explanation of the function. There is an option to switch these off in the “Display Options” settings when you get used to the functions.

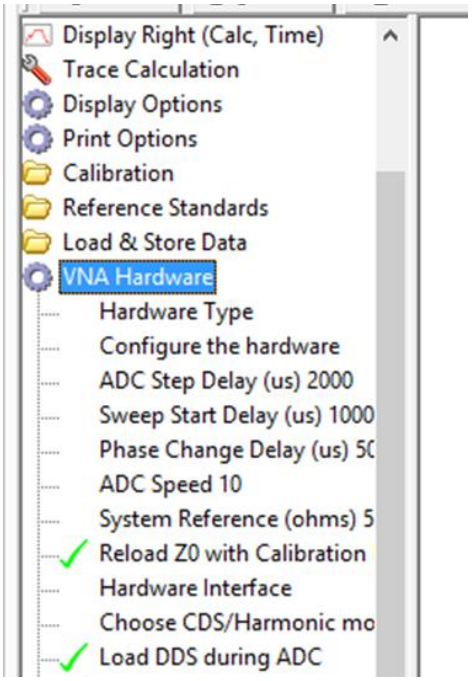
As the program knows you have no calibration data it has opened up the calibration data controls, but you need to set the hardware up first. Experiment with this tree control. You will find that as you click on each of the main items, it will open up and the previous one will collapse to keep the display manageable.

4.2 Setting Initial Hardware Settings

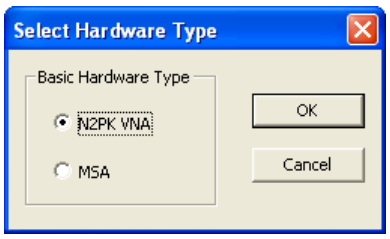
Before anything else, the hardware settings need to be configured. Select the “VNA Hardware” node. The display will change to show you the controls as shown below:



First, select the Hardware Type from the option under the “VNA Hardware” option of the tree control as shown below:



This opens a dialog box as shown:

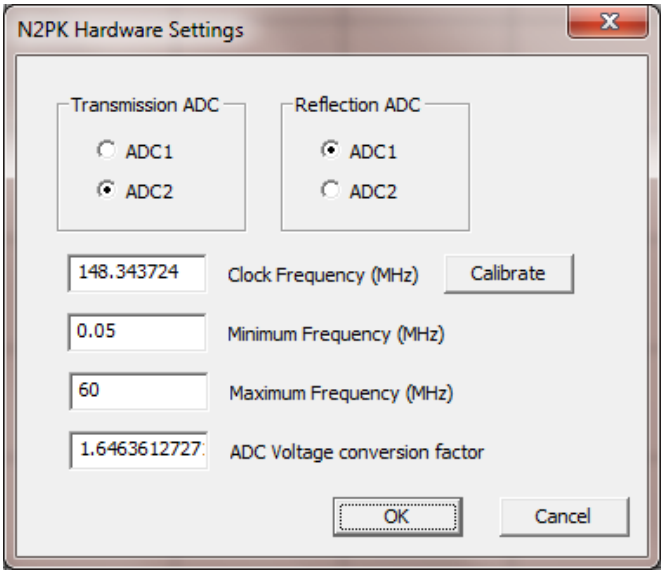


Select the desired hardware type and click “OK”.

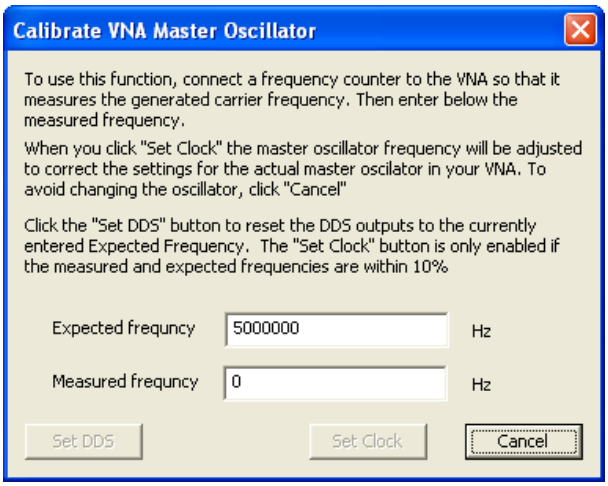
4.2.1 N2PK Hardware Setup

Next configure the hardware. For the N2PK this is easy – just select the "configure the hardware" option under "VNA Hardware" in the treeview to the left.

.



Enter the desired settings and click OK. If you wish to set the master clock frequency, there is a Calibration function to help. Click the "Calibrate" button, but note that you can't do this until you have selected the hardware interface, so come back to this later…
To set the master clock frequency, first make sure the VNA has warmed up and connect a digital frequency meter to the RF output.


Calibrate VNA Master Oscillator
To use this function, connect a frequency counter to the VNA so that it measures the generated carrier frequency. Then enter below the measured frequency.
When you click "Set Clock" the master oscillator frequency will be adjusted to correct the settings for the actual master oscilator in your VNA. To avoid changing the oscillator, click "Cancel"
Click the "Set DDS" button to reset the DDS outputs to the currently entered Expected Frequency. The "Set Clock" button is only enabled if the measured and expected frequencies are within 10%
Expected frequncy
5000000
Hz
Measured frequncy
0
Hz
Set DDS
Set Clock
Cancel

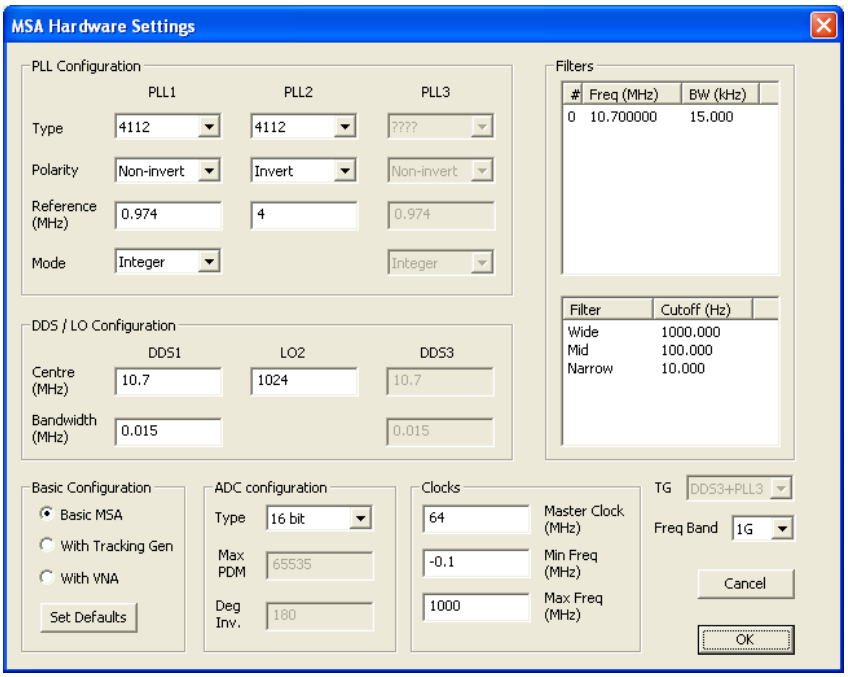
The frequency counter should display a frequency close to the expected frequency shown. Enter the measured frequency. If it is within 10% of the expected frequency (this is just to prevent "finger trouble") then the "Set Clock" button will be enabled and, if clicked, will close the dialog and change the master clock frequency to calibrate for the oscillator in the VNA hardware.

The "default" frequency shown here is 5 MHz. As soon as this dialog is opened, the program will attempt to set both DDSs to that frequency. If you wish to use a different

frequency, enter it in the “Expected Frequency”. The “Set DDS” button will then be enabled and when clicked it will set both DDSs to the new frequency.

The final entry, “ADC Voltage conversion factor” should not normally need to change. It is there for anyone who uses myVNA to control other hardware. In this case, the ADC reading is multiplied by this conversion factor to convert the value into volts. The value shown here is 1.64636127271944e-009, which is the ADC hardware reference voltage (2.5) divided by 2^{28} as the drivers internally use a 28 bit reading, divided by $2*\sqrt{2}$ for rms conversion.

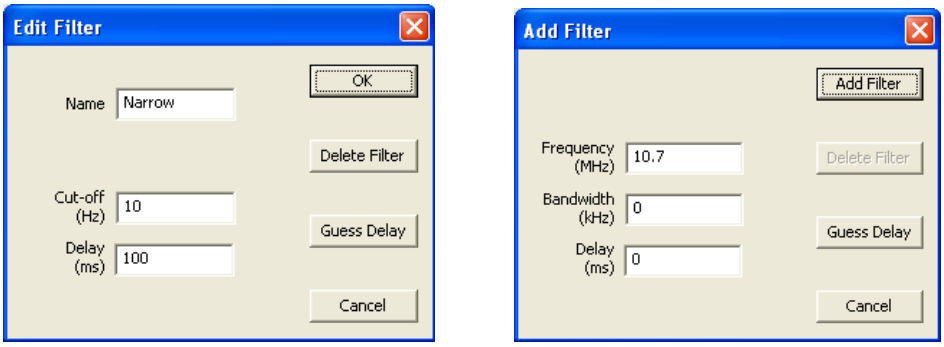
4.2.2 MSA Hardware Setup



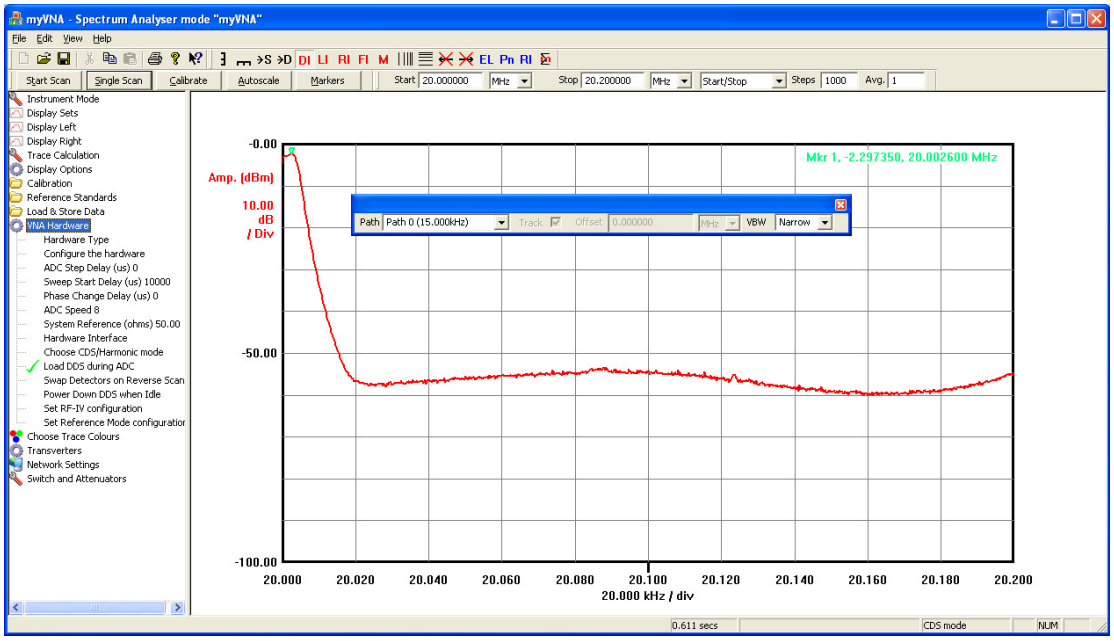
The first time you run it, the PLL types will show as “????”. Select the basic MSA type from the “Basic Configuration” options bottom left then click “Set Defaults”. Now change the options to match your hardware.

The filters are set using the two filter edit windows on the right. To change the entries, either double click with the mouse or right click for a menu. If you double click on an entry, it opens a dialog to edit the entry, double click away from an entry and it offers the chance to add an entry.

The top pane configures the 10.7 MHz band pass filters. The lower pane configures the video low pass filters. Set each as needed but – importantly – set the delay value. If you are not sure click the “guess” button once the bandwidth has been entered. Here are a couple example dialogs.



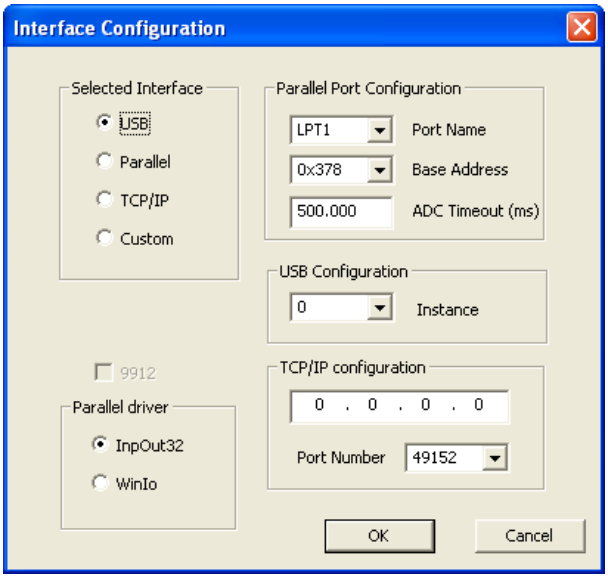
On the main screen of the program is a new toolbar not used for the N2PK VNA. See the figure below. Like all toolbars here, it is a dockable, floating toolbar and can be moved around to where you want it.



Select the video filter and path. From “Instrument mode” select “Spectrum Analyser” and from the Display Left select “Amp. (dBm). Untick any option in “Display Right”.

4.3 Hardware Interface Selection

Select the “Hardware Interface” option from the “VNA Hardware” section of the tree control. This pops up a dialog box like this



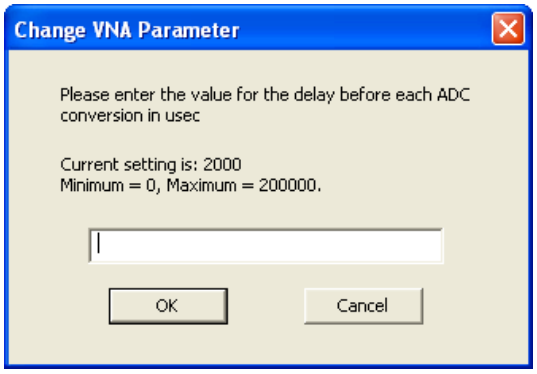
Select the interface type you are using (MSA only supports USB) and if parallel is chosen, select the base address of the printer port and the port name. The TCP/IP option is described in Section 33 on page 38.

For the parallel port, the address is the one used for I/O; the port name is used simply to reserve the port and try to stop other programs from accessing it. Note that if you want an address different to that in the drop down list, just type in your desired address. The program enforces basic limits on the address but does not otherwise check its correctness. If however you select a "non-standard" address it will ask you to confirm it. If you enter a silly value and confirm it, the program will try to use it. If that happens to be some other I/O device then you are liable to crash your computer. Take care.

You have a choice of parallel drivers to choose from. Start with InpOut32. This is the slower option. Use it initially and, once you are certain all works correctly, try WinIO. It should be faster. Remember that under Vista you must have local admin rights if WinIO is used.

The Custom interface uses a defined API from a custom DLL and by this provides a means to access other hardware VNA designs provided that a suitable interface DLL is created.

Now change the other parameters as needed.

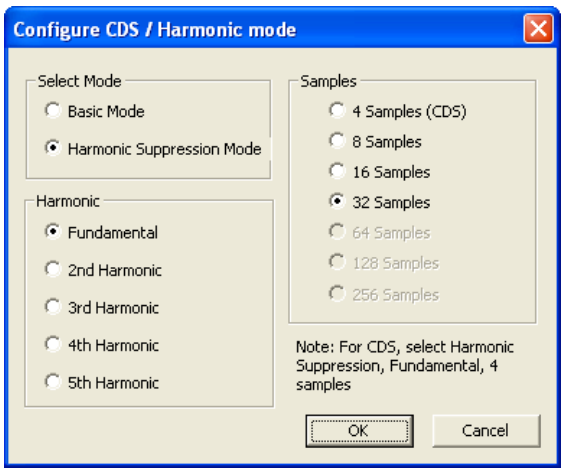

Change VNA Parameter
Please enter the value for the delay before each ADC conversion in usec
Current setting is: 2000
Minimum = 0, Maximum = 200000.
OK
Cancel

When you click on a control with a value associated with it such as the ADC step delay, a dialog will open up allowing you to change the value, as shown below. The min and max frequency parameters relate to the frequency range of the base hardware VNA. The default is 50 KHz to 60 MHz but change it as you wish. All settings are retained when you exit & restart the program.

Ignore the "USB Instance" parameter for now. This is designed to allow multiple USB connected VNAs to be used at the same time on one PC. For more details, please refer to Section 31on page 126.

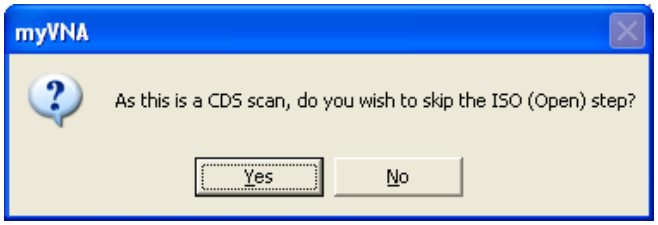
4.4 Choose CDS / Harmonic mode

This treeview node from the VNA Hardware section allows the harmonic mode to be selected. In Basic mode, a single pair of measurements is made at each point, one reading 90 degrees out of phase with the other. Harmonic suppression modes perform additional measurements and also permit selection of harmonics of the DDS frequency. The dialog box is as shown below:



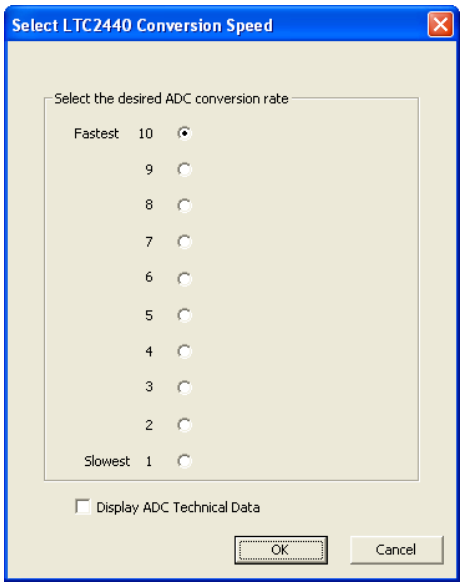
For now select "Harmonic Suppression Mode" with "Fundamental" and "4 Samples (CDS)". This is also known as Correlated Double Sampling (CDS) mode. There is more on harmonic modes later in this manual.

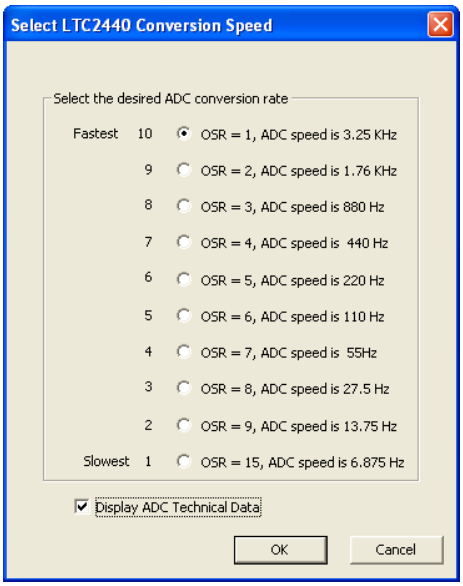
Also, when calibrating in transmission mode with a CDS sweep it will offer you the option to skip the ISO (Isolation aka "open") calibration as shown below:



4.5 Setting the ADC mode

The ADC mode is shown as a "speed" value from 1 to 10 with 1 the slowest, and 10 the fastest. When you change the ADC speed (to be found under the "VNA Hardware" treeview node), a dialog pops up as shown below. Select the desired ADC mode by selecting the appropriate radio button then clicking OK. Note that two forms of display are possible; technical ADC data is shown if you click the "Display ADC Technical data" box. Both forms are shown below, and the program assumes both ADCs are of the same type.




Select LTC2440 Conversion Speed
Select the desired ADC conversion rate
Fastest 10 OSR = 1, ADC speed is 3.25 KHz
9 OSR = 2, ADC speed is 1.76 KHz
8 OSR = 3, ADC speed is 880 Hz
7 OSR = 4, ADC speed is 440 Hz
6 OSR = 5, ADC speed is 220 Hz
5 OSR = 6, ADC speed is 110 Hz
4 OSR = 7, ADC speed is 55Hz
3 OSR = 8, ADC speed is 27.5 Hz
2 OSR = 9, ADC speed is 13.75 Hz
Slowest 1 OSR = 15, ADC speed is 6.875 Hz
Display ADC Technical Data
OK
Cancel

4.6 Setting the Display Options

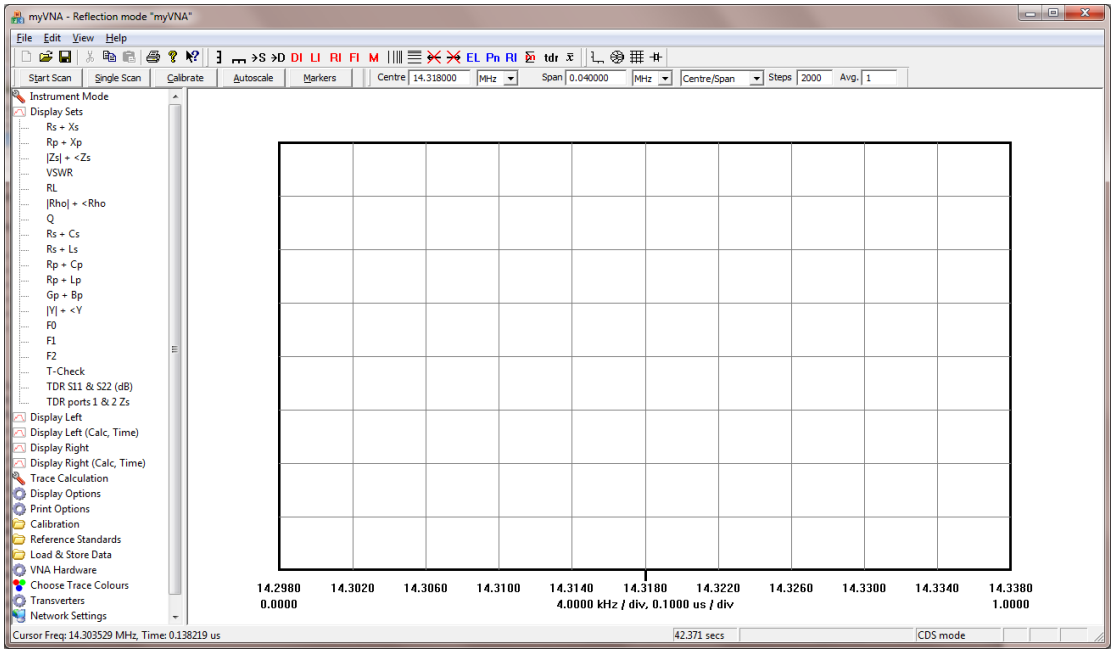
Select the Display Options from the main tree view and change these as needed. Note the Graticule setting. A green tick should be shown. Items that are either selected or not just gain or lose a tick mark when you click them.

You have the option to switch on or off a scan progress bar. This appears on the status bar below the sweep display and shows the progress of the current sweep.

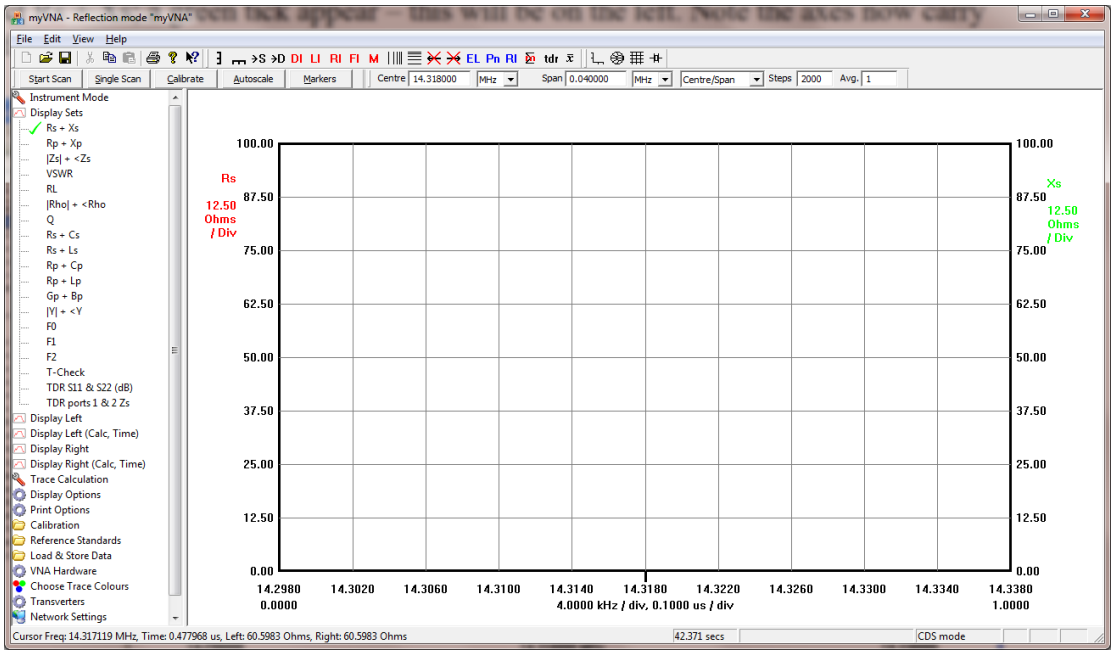
Also on the status bar will appear the expected scan run time. This is based on how long things take on a typical PC and may be different for other systems, use it as a rough guide only.

4.7 Performing your first scan

Assuming all of the above settings have been made correctly, go to Instrument Mode and select either Transmission or Reflection. For whatever the instrument mode is, the Display Sets will reflect the current mode. Try changing them and see what happens. In general you have one axis on the left, one on the right and a frequency axis on the bottom. Whenever you select a new left display item any previous one is removed. To get the idea of this, select "Reflection" mode then open up display types like this



Select Rs + Xs; a green tick appear – this will be on the left. Note the axes now carry a default setting and labels of Rs and Xs.



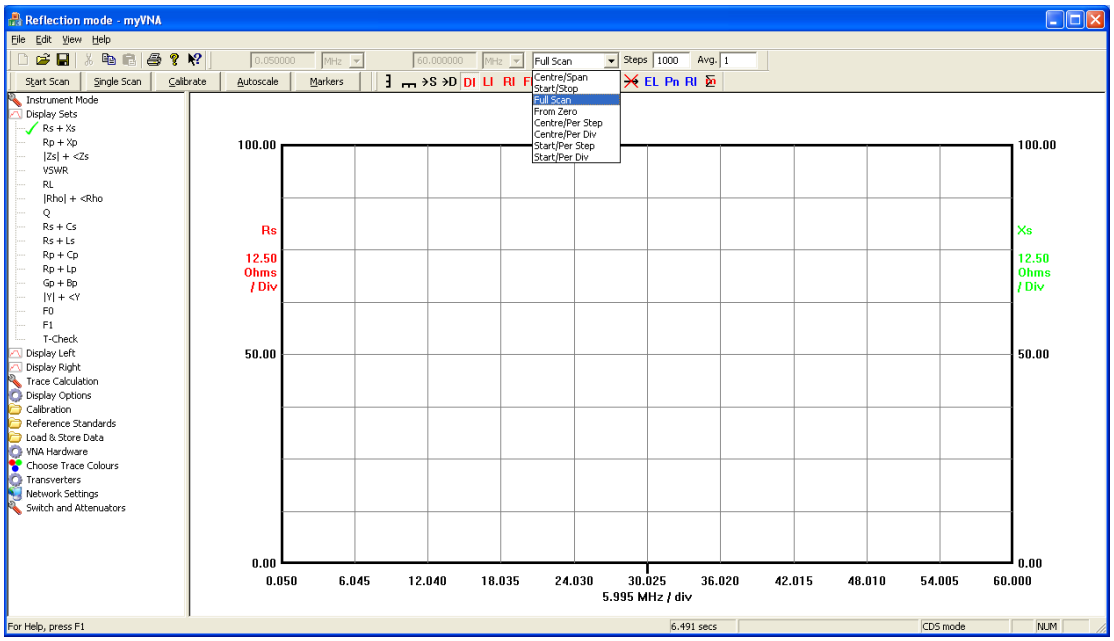
Now select Rs + Cs; Cs will replace Xs.

Now look again at the TreeView. Beneath the currently displayed “Display Sets” are options “Display Left” and “Display Right”. These allow you to individually select what parameter to show on the left axis and what on the right. This is for the occasions when you don’t want the predefined groupings of “Display Sets” and want to choose them independently. Your choice. You can hide the sets option if you want to. Incidentally, as will be described later on, you may display multiple data types at the same time by selecting multiple parameters, holding down the CTRL key and clicking on the desired choices. Finally, whilst it is usually the case that the offered

choices will match the needs of the instrument setting, you may find that you want all choices available at the same time. To switch the program into this mode, open up the options dialog from the View menu at the top of the screen and tick or untick the option as necessary.

OK, select Rs + Xs again from the “Display Sets” option.

On the span toolbar, change the mode to “full scan” from the drop down list

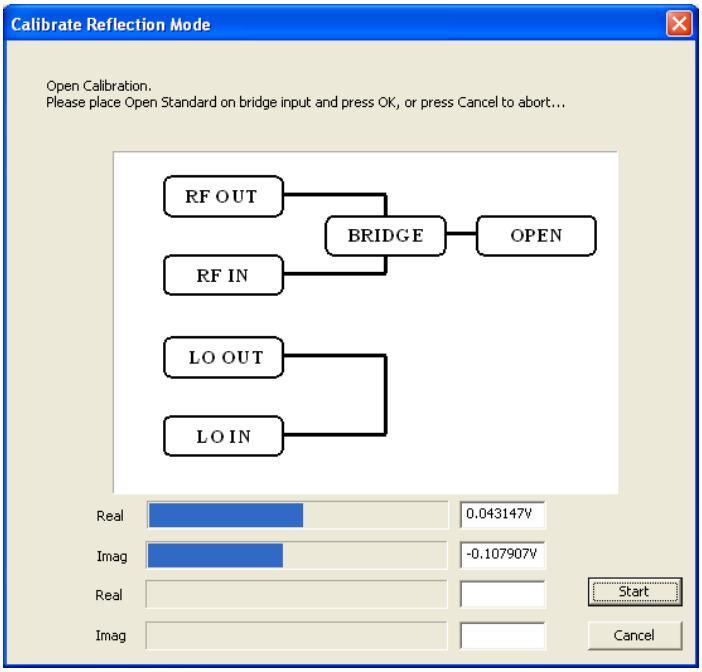


The two frequency controls will be greyed out as it will now do a full scan if requested.

Also present under “Display Options” are a couple of entries that control the labelling of the horizontal and vertical axes in rectangular view mode. When ticked, these cause the respective axis to be labelled at every graticule point and not just the first, last and central ones. If you select the option to label the vertical axes, you may want to drag the display type and units label away from the axes value labels so they do not overlap.

4.7.1 Calibration

Click the “calibrate” button and follow the instructions starting with this



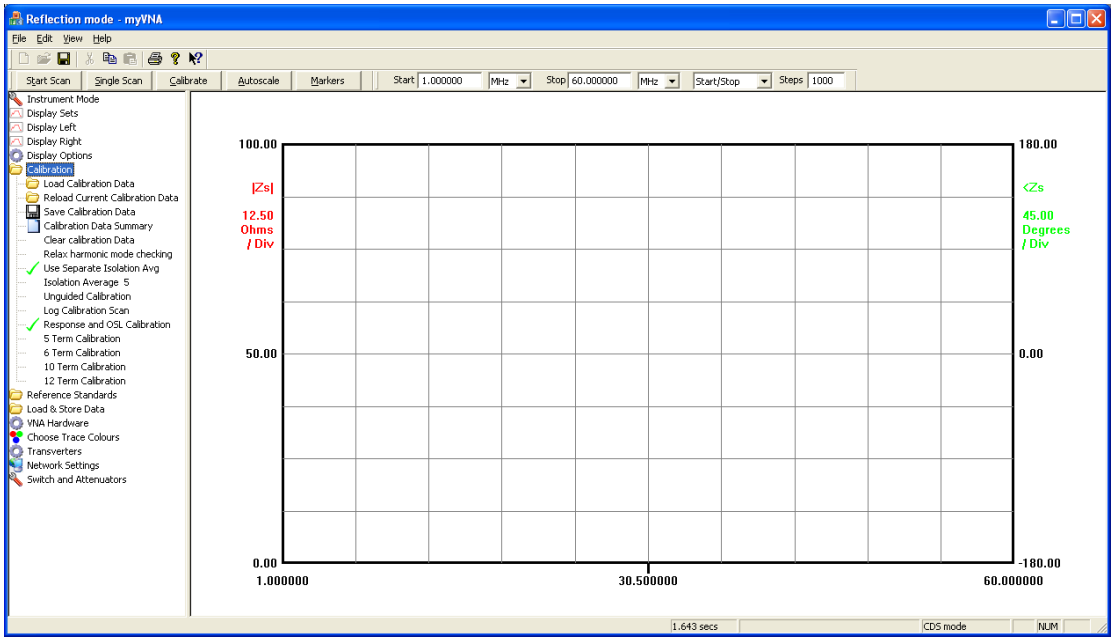
The display shows a simple picture of the expected connections and a set of progress bars that will display the current readings for the ADCs, continually updated. In reflection or transmission mode, just the first pair is used. For dual mode scan, both are active. The voltage of each is also displayed on the right of the progress bars. If you get nothing displayed in the progress bars and no voltages shown, it means that the hardware is not responding correctly to the software.

Assuming you get no error messages it will ask you for open, short and load and finish with a 'save file' dialog. Choose a suitable name for the calibration data – the program will automatically remember this filename and load the calibration data at startup next time (along with all the other parameters you just set). It is suggested that you store the calibration data in the folder "myVNA" under "My Documents". This folder should have been created for you when the program was installed.

As long as the calibration data is valid for the scan (i.e. the scan frequency is not outside the calibration data) it will permit the scan, interpolating the calibration data for the scan points selected if necessary. Also the scan data will hold two sets of data at the same time, one transmission and one reflection – PROVIDED that the frequencies and number of step points are the same. It permits this as part of its support for dual detector scans and S parameters.

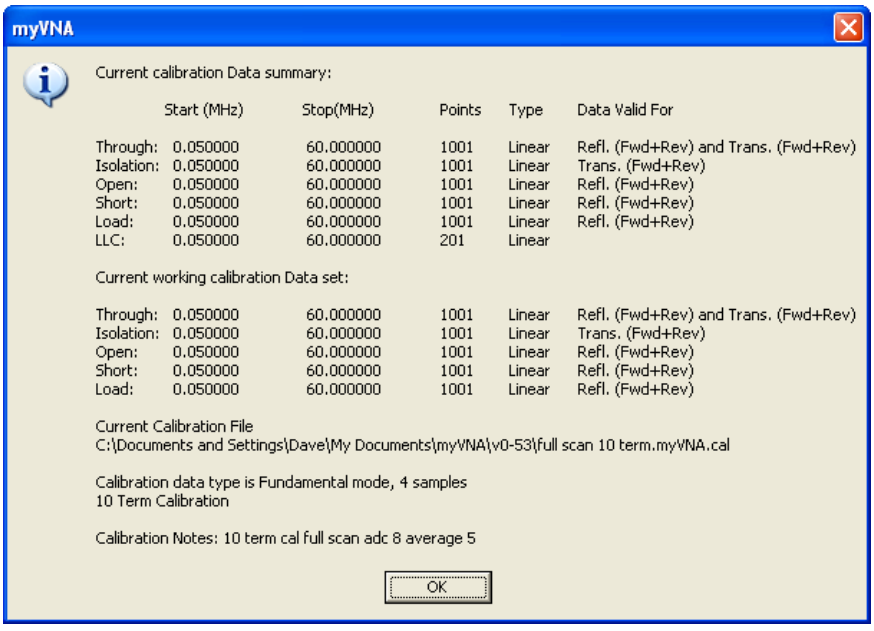
Note that the program will not allow you to mix CDS and non CDS calibration data. Similarly it will not allow mixing of harmonic modes unless you explicitly override this limit for test purposes. There is a flag to permit this under the "Calibration' node.

Also you can choose to use additional averaging on the isolation step of CDS. There are two options under "calibration" to support this as shown below. When selected (as shown), the average number in the following line is used for the isolation step average count instead of the normal display average count (under "display options"). The number of averages at each point may also be set separately for calibration as for scans at this point.



The form in which calibration data is stored is in its raw format, without calibration standard offsets applied. This is to allow steps in the calibration process to be repeated. When it is used, it is corrected. This also applies to when it is loaded from disk. Therefore if you change the calibration data standard settings then reload calibration data, it will be corrected for those new settings.

Finally on calibration, you can display a summary of the current calibration data from the Calibration node on the left pane. It will result in a dialog box something like this. The filename of the current calibration file is shown together with the details. If the file contains any calibration notes these are shown at the bottom of the dialog.

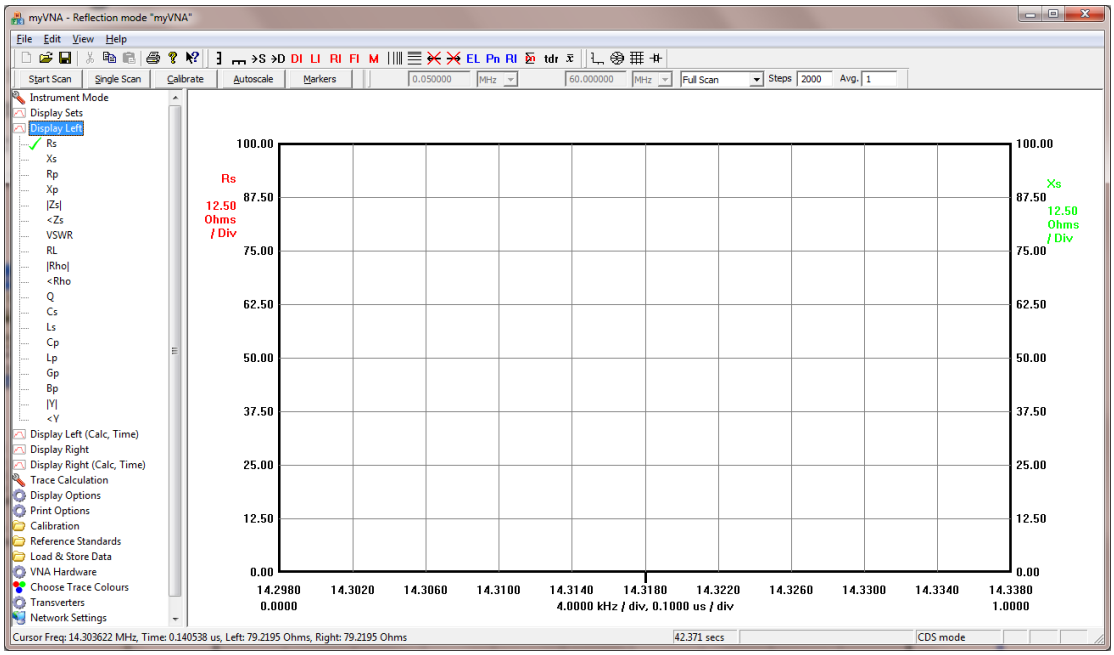

myVNA
Current calibration Data summary:
Start (MHz) Stop(MHz) Points Type Data Valid For
Through: 0.050000 60.000000 1001 Linear Refl. (Fwd+Rev) and Trans. (Fwd+Rev)
Isolation: 0.050000 60.000000 1001 Linear Trans. (Fwd+Rev)
Open: 0.050000 60.000000 1001 Linear Refl. (Fwd+Rev)
Short: 0.050000 60.000000 1001 Linear Refl. (Fwd+Rev)
Load: 0.050000 60.000000 1001 Linear Refl. (Fwd+Rev)
LLC: 0.050000 60.000000 201 Linear
Current working calibration Data set:
Through: 0.050000 60.000000 1001 Linear Refl. (Fwd+Rev) and Trans. (Fwd+Rev)
Isolation: 0.050000 60.000000 1001 Linear Trans. (Fwd+Rev)
Open: 0.050000 60.000000 1001 Linear Refl. (Fwd+Rev)
Short: 0.050000 60.000000 1001 Linear Refl. (Fwd+Rev)
Load: 0.050000 60.000000 1001 Linear Refl. (Fwd+Rev)
Current Calibration File
C:\Documents and Settings\Dave\My Documents\myVNA\v0-53\full scan 10 term.myVNA.cal
Calibration data type is Fundamental mode, 4 samples
10 Term Calibration
Calibration Notes: 10 term cal full scan adc 8 average 5
OK

There are two sets of data shown. The first set is the “Raw” calibration data as currently held from a calibration scan. The program generates a per scan set of data from this raw data, and the second set of data indicates the current scan specific calibration data. In each set, 5 rows are shown. The Through Data indicates the transmission (thru) data and also the reflection data of the second detector when used for ERC, 10 or 12 term correction. The Isolation row shows the transmission Isolation data. The Open, Short and Load rows show the reflection data for OSL correction.

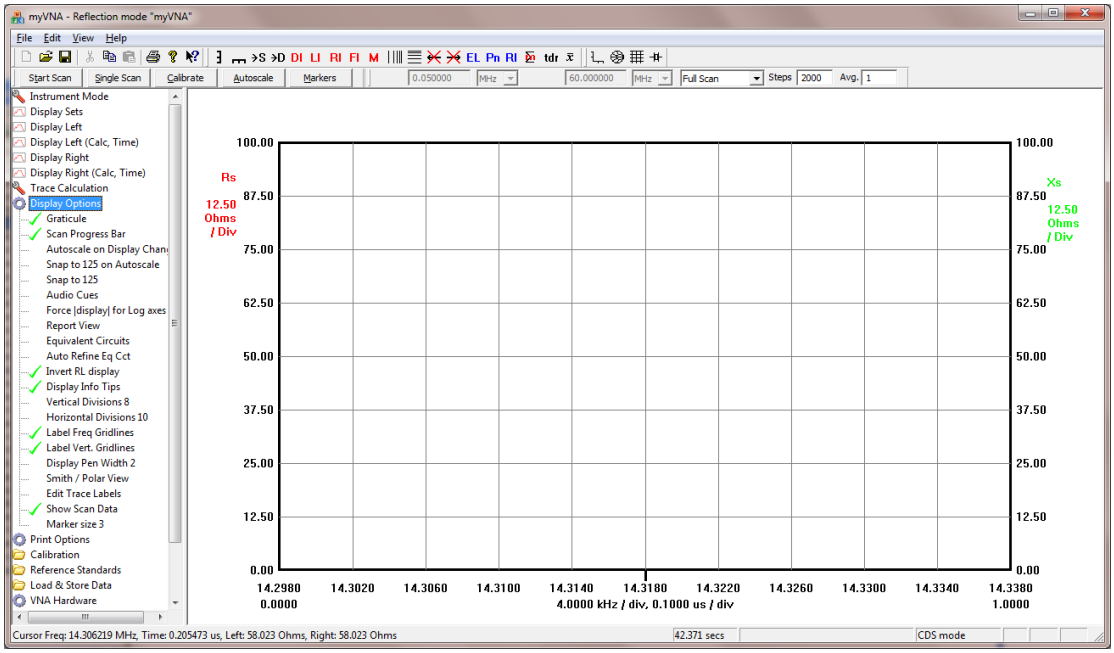
The Type indicates whether the calibration data was created using a logarithmic or linear step frequency scale. Although you may freely mix log or linear calibration data with log or linear device scans, it may be better, especially at extremes of frequency, to use log calibration data with a log device scan and vice versa. From 1 to 60 MHz unless you use a small number of points it makes no difference.

4.7.2 Scanning

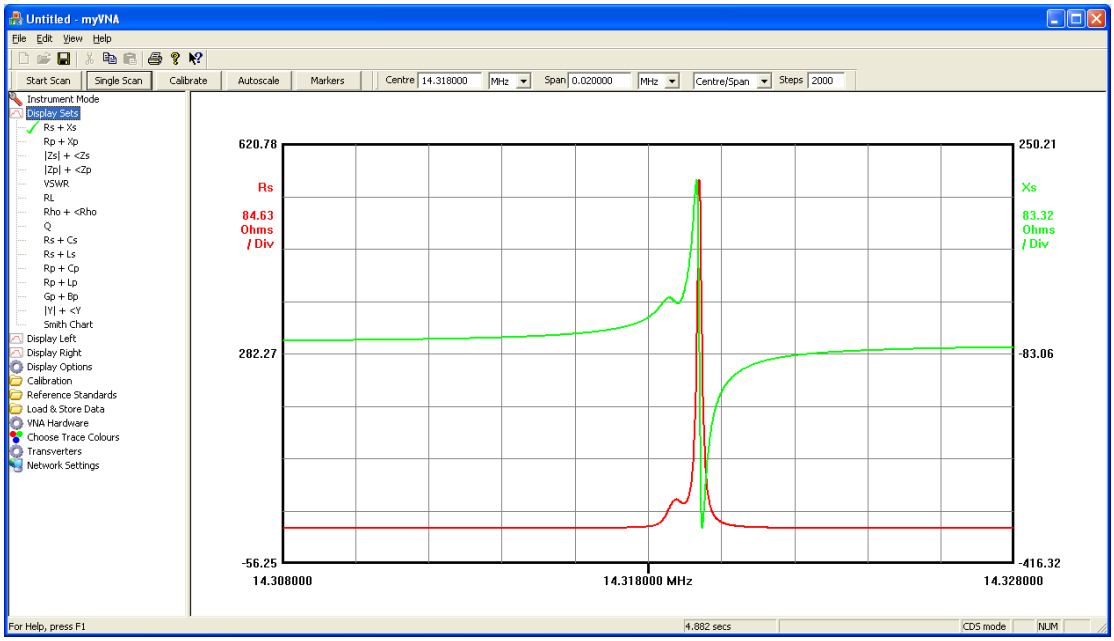
The scan is set using the span bar controls – select centre and span from the drop down list, apply something interesting and set the scan, for example as shown here.



Now, before you do a scan, open up the “display options” node on the left and make sure that “autoscale on display change” and both “snap to 125” options are ***not*** selected as shown below. Note in this example, the option to display Info Tips is selected.



Click “Single Scan” and wait. It is likely that the display will be off-screen. On the first trace, it will autoscale for you, but in general clicking “Autoscale” will rescale the axes to match the current trace. When you try dragging things around later, click autoscale to reinstate the axes.

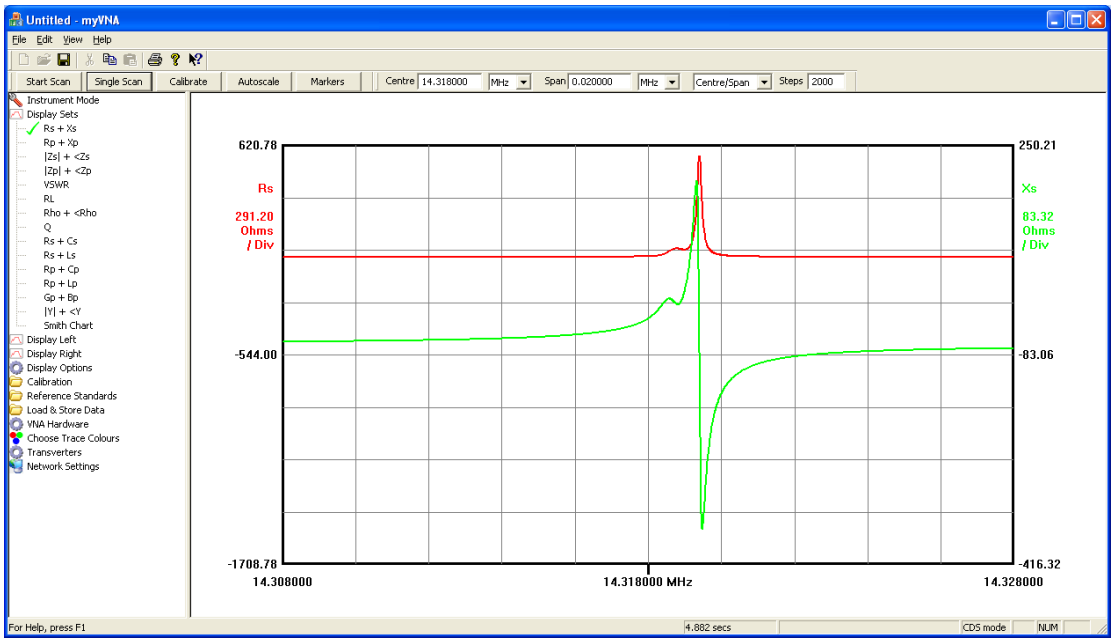


4.7.3 Dragging the display around

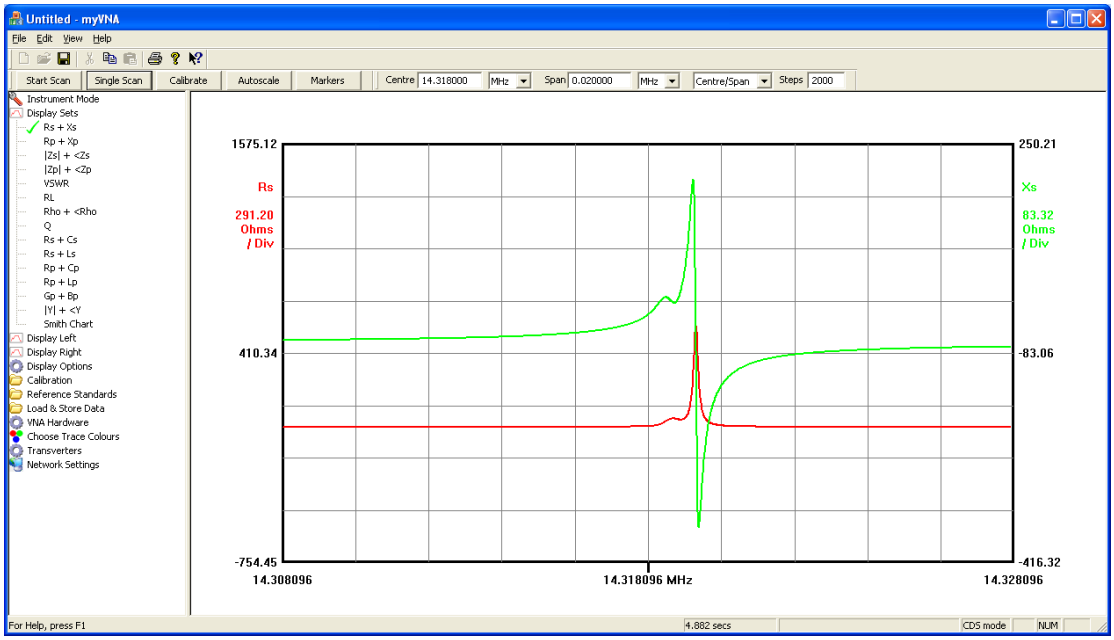
Autoscale brings both main traces on screen. It will also take account of any stored trace data *currently displayed* and ensure that both the stored data and current trace fit on screen.

Now time to play with dragging things. There are 6 different draggable things here, 5 to be described now.

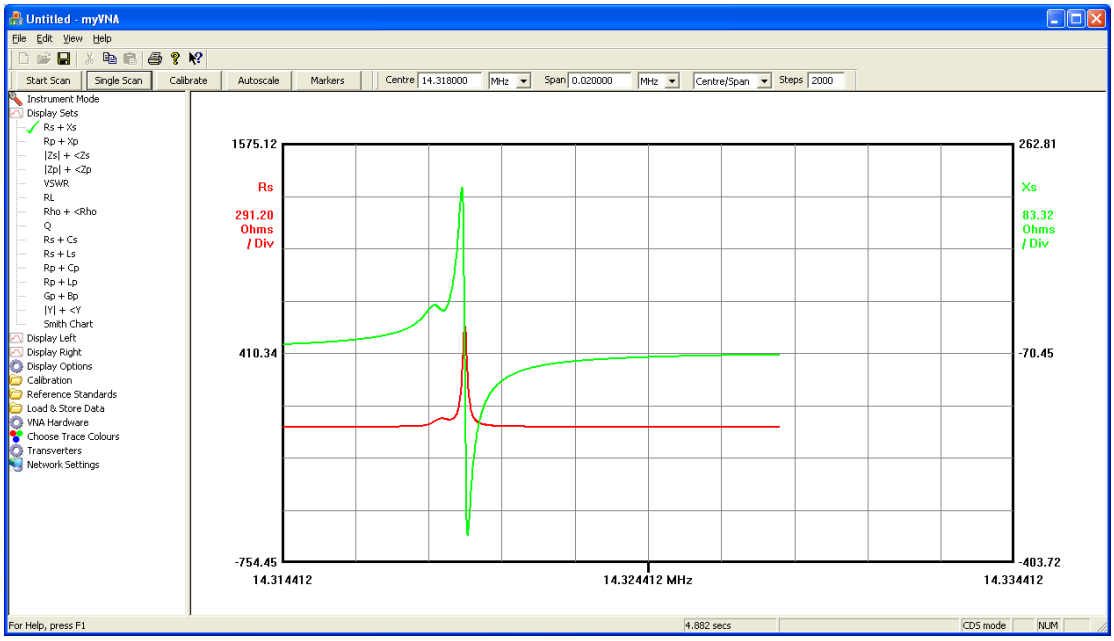
On the left of the left vertical axis and right of the right vertical axis are zones where the axes may be rescaled. Put the mouse on the left say, somewhere between the values (in the example above 139.21 and -27.30. Press the left mouse button and drag up or down. The value at the top of the axis will remain unchanged but the scale will increase or decrease – for example moving the mouse up gives this



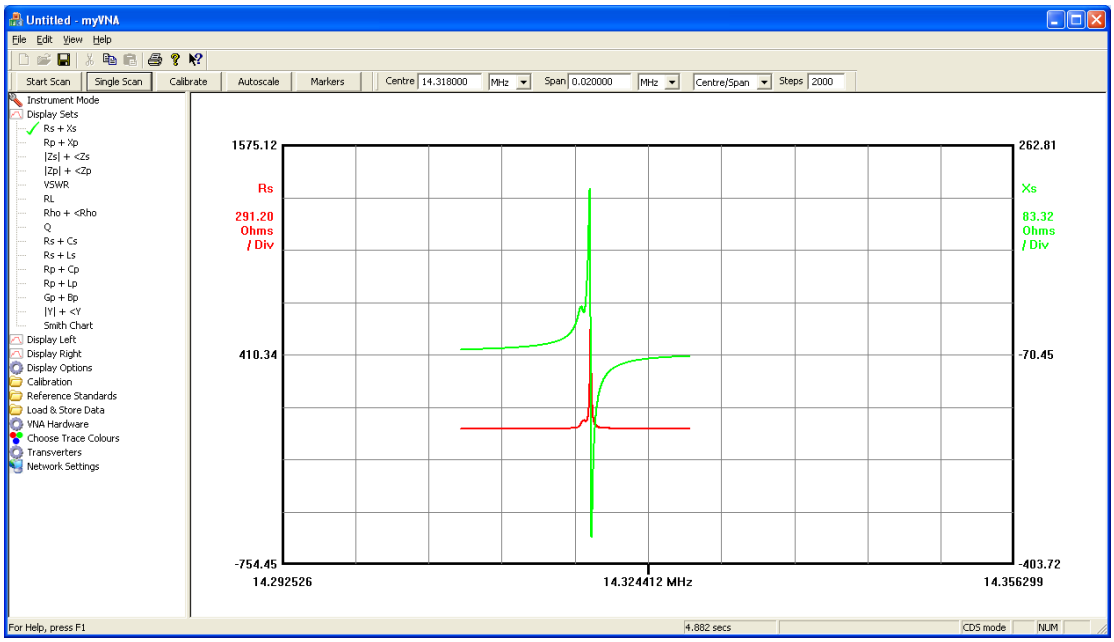
If you drag within the graticule it will move the trace up or down; specifically the left axis trace if you click and drag on the left half of the graticule or the right trace for the right half. This is the result of dragging the left axis trace down



Note also that the frequency axis has moved in this example. This is because you can also drag the frequency axis left and right like this



If you drag below the frequency axis then it will increase or decrease the span, for example



Note that this drag is temporary – the frequencies in the span bar are unchanged to allow you to return to it easily. Before doing that though, do another single scan – it will use the temporary frequency range you selected by dragging. If you want to change the span bar settings to match the frequency scale you have created by using the mouse, right click in the display area and select the menu option “Set Scan to Current Frequencies”

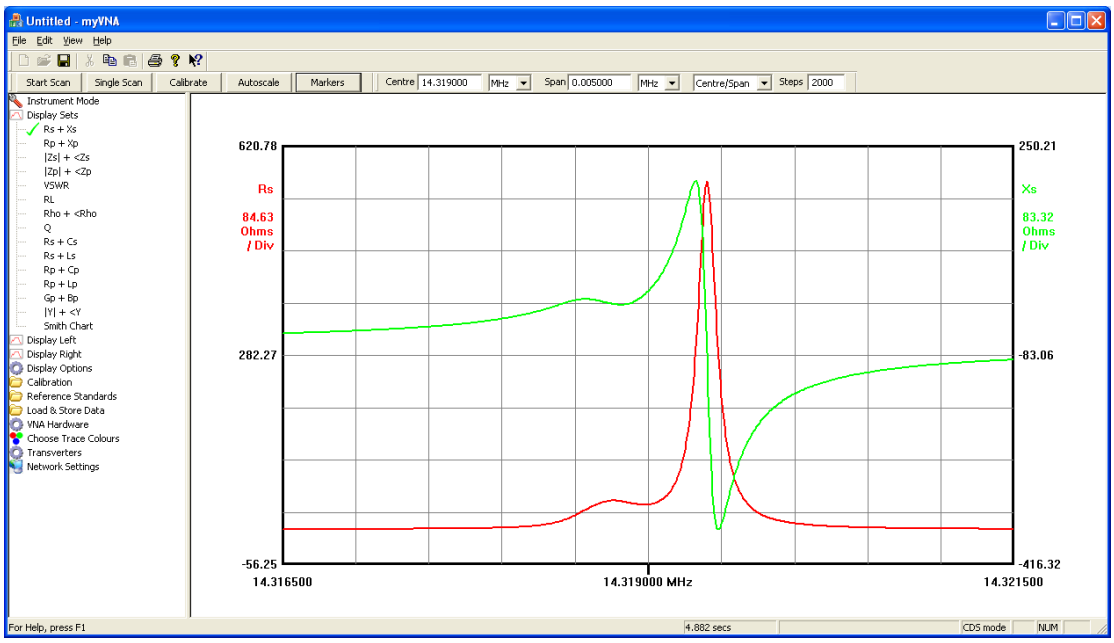
Now open up the “display options” tab on the left again and click the “Autoscale on Display Change” option. Try changing the Display Type. Now, each time you change the display (but not every time it completes a sweep), the display will autoscale.

Look again at the “display option”. There are two marked “Snap to 125”. These will avoid those strange units you noticed as you dragged traces around – basically at the moment you may drag traces smoothly. If you select a 125 snap option, the display will try to assume nice round numbers for the units on a 1/2/5/10 type scale when you release the mouse button. It will do this separately for each drag option, and you may also select for this to happen on autoscale. Personal preference.

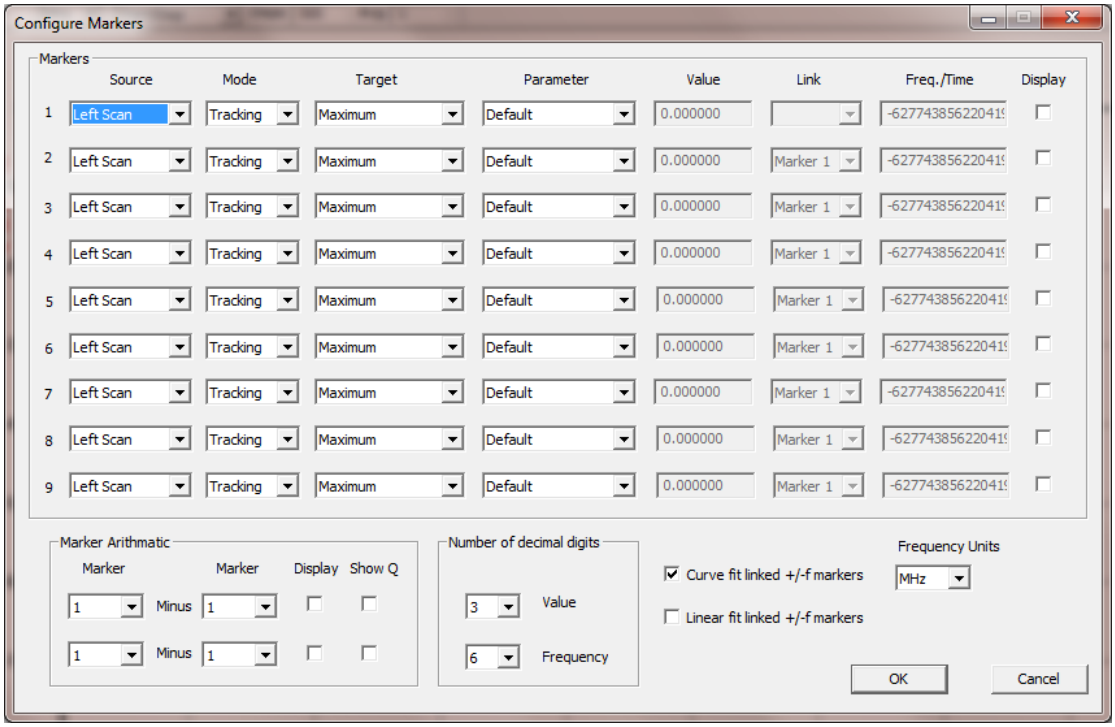
Now try a repeat scan with the “scan” button. You can drag the display around and rescale it whilst it is scanning but the display you get if you drag the frequency is a bit weird until it gets chance to do a complete scan and sort the display out. If this is not to your liking there is an option to disable dragging of the frequency scale whilst scanning – see the section “Program Limits and other options” a few pages on from here.

4.7.4 Markers and Marker Arithmetic

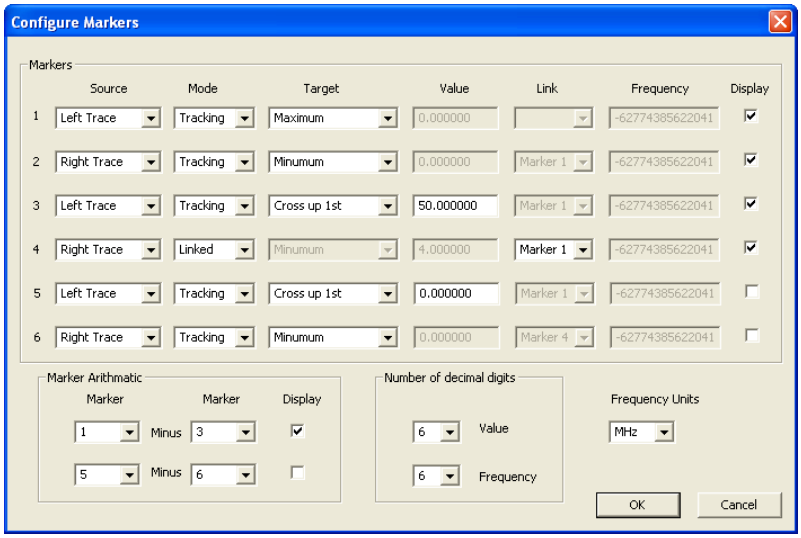
Make a display with some peaks and dips on it such as the one shown



Now click the “Markers” button. You get a display like this

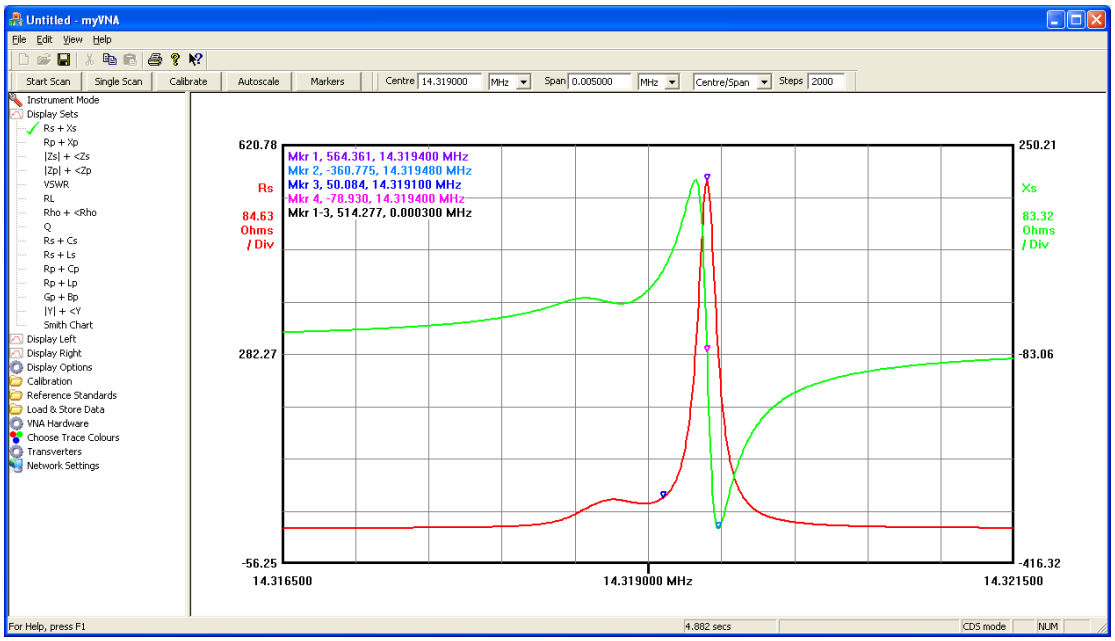

Configure Markers
Markers
Source Mode Target Parameter Value Link Freq./Time Display
1 Left Scan Tracking Maximum Default 0.000000 -627743856220419
2 Left Scan Tracking Maximum Default 0.000000 Marker 1 -627743856220419
3 Left Scan Tracking Maximum Default 0.000000 Marker 1 -627743856220419
4 Left Scan Tracking Maximum Default 0.000000 Marker 1 -627743856220419
5 Left Scan Tracking Maximum Default 0.000000 Marker 1 -627743856220419
6 Left Scan Tracking Maximum Default 0.000000 Marker 1 -627743856220419
7 Left Scan Tracking Maximum Default 0.000000 Marker 1 -627743856220419
8 Left Scan Tracking Maximum Default 0.000000 Marker 1 -627743856220419
9 Left Scan Tracking Maximum Default 0.000000 Marker 1 -627743856220419
Marker Arithmatic
Marker Marker Display Show Q
1 Minus 1
1 Minus 1
Number of decimal digits
3 Value
6 Frequency
Curve fit linked +/-f markers
Linear fit linked +/-f markers
Frequency Units
MHz
OK
Cancel

There are 9 markers and two marker maths functions. For this example, choose to display marker 1 which tracks the maximum of the left trace, Marker 2 that tracks the minimum of the left trace, marker 3 that tracks the first time that the right trace crosses 50 ohms rising, and marker 4 that is going to track marker 1 but is on the right trace. Depending on what you scanned, you may need to alter this. Don't forget to tick the 'display' box for each marker that you want to enable, but to help you, whenever you change the source mode or target settings of a marker, the marker will be enabled (the "display" box automatically ticked) if it was not enabled. Please note the following screenshots use an earlier version that had 6 markers.


Configure Markers
Markers
Source Mode Target Value Link Frequency Display
1 Left Trace Tracking Maximum 0.000000 -62774385622041
2 Right Trace Tracking Minumum 0.000000 Marker 1 -62774385622041
3 Left Trace Tracking Cross up 1st 50.000000 Marker 1 -62774385622041
4 Right Trace Linked Minumum 4.000000 Marker 1 -62774385622041
5 Left Trace Tracking Cross up 1st 0.000000 Marker 1 -62774385622041
6 Right Trace Tracking Minumum 0.000000 Marker 4 -62774385622041
Marker Arithmatic
Marker Marker Display
1 Minus 3
5 Minus 6
Number of decimal digits
6 Value
6 Frequency
Frequency Units
MHz
OK
Cancel

Note also that the number of digits to the right of the decimal point may be set for the markers using the drop down list. There are separate settings for the value and the frequency.

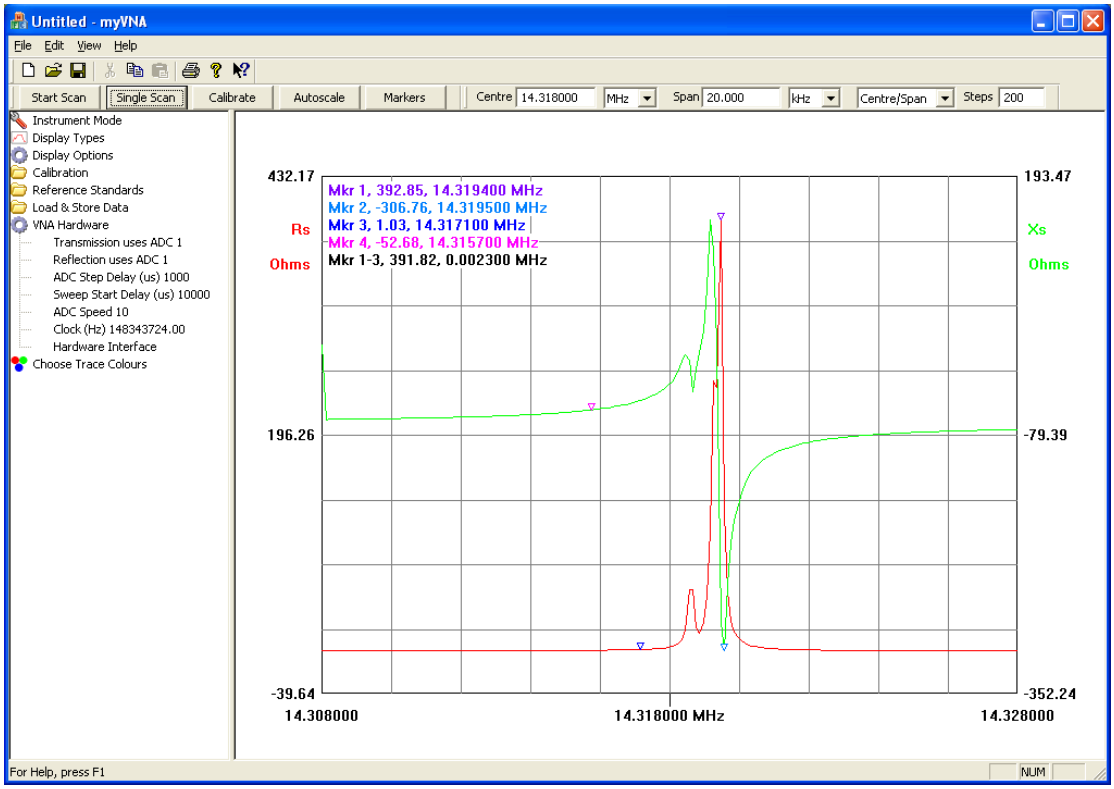
Also ask for it to display the difference between markers 1 and 3. The display now looks like this



Ask for it to scan repeatedly. The markers will be updated at the end of each scan tracking what they are asked to track.

When you perform marker arithmetic, the program will not stop you subtracting, say, degrees from ohms. It is up to you as to whether the units make sense. Also note that if the value of a marker is computed as being over 1e9, the marker will not display.

Note that all the images shown in this manual apart from the next one are on a 1660x1050 pixel widescreen display. Here is an example of a display on a 1024x768 monitor



Now go back to the markers dialog and change one of the markers to Manual. Set its frequency to be somewhere in your scan (note the frequency may be selected to be in Hz, KHz or MHz). Close the dialog. That marker is no longer a tracking marker, it will stay put on its frequency, but now you can also drag it (this was the other draggable item referred to previously). If you put the mouse pointer close to it (technically, within 10 pixels) and depress the left mouse button, you can drag it left and right. The marker will stay glued to its trace but can now be moved left & right.

Look at the status bar as you drag the manual marker – it will indicate the current marker position and values. This only lasts whilst the marker is being dragged. When you move the mouse across the screen at other times, the current mouse pointer position will be displayed as frequency & values for left & right axes.

Markers are by default positioned top left of the display. Sometimes this is not very convenient – it obscures parts of the trace. So you may move it around on the screen by dragging it where you want it to be. Anywhere on the marker display text, click the left mouse button and drag the markers where you want them to be. The position you select will be preserved on startup, and there are two positions, one for polar/smith charts and one for rectangular. Finally, there are a few quirks; the position may be off the page on printed copies and dragged off the bottom they may reappear on the top as they strive to stay visible.

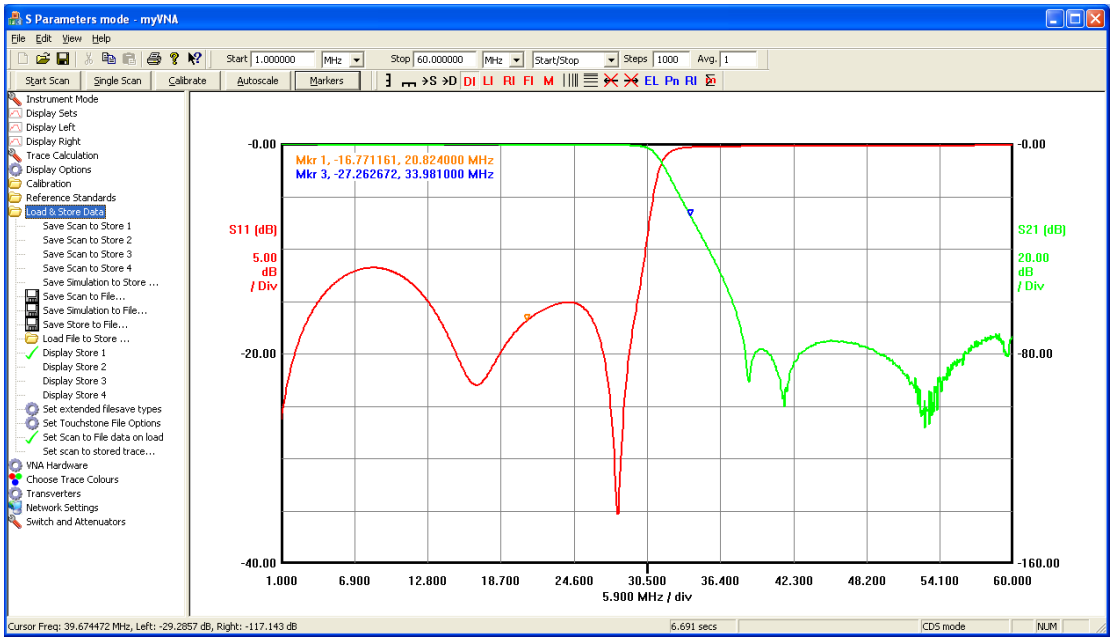
In Polar and Smith Chart views, the markers work slightly differently. At their core, they always use the modulus and argument of the measurement. Therefore when setting a marker, the “left” trace refers to the modulus and the “right” trace refers to the argument. Also in Smith Chart view, the values that the markers display may be changed to another parameter, for example Rs+Xs. Underlying this though, the program is still dealing in rho, so the values used in tracking markers will still be rho.

4.8 Data Stores

Now select Load & Store Data and click on Store 1. The current scan data will be stored in the first of 4 data stores and displayed. The display can be toggled on or off with the “Display Store 1” item as shown below.

The basic model is as follows. You can have 4 data stores that hold trace data, and can display one or more of these with the current scan data. The current scan data can also be stored to a file and a set of stored data in a file can be loaded to one of the stores. If you load stored data without a scan trace, you may display, analyse and view the stored data as if it were from a recently completed scan.

Whenever you have a stored trace on screen, autoscale will also take it into account when it autoscales the display.



Note that the colours have changed as the stored data is written after the scan data and for now is identical.

You can choose different colours with the leftview controls

Now, once you have a set of data from a scan such as that shown above, you can change the way that it displayed without rescanning using the “display types” control. Try selecting something else and, if needed, autoscale.

You may save data into any combination of the 4 stores and display any combination of them at the same time.

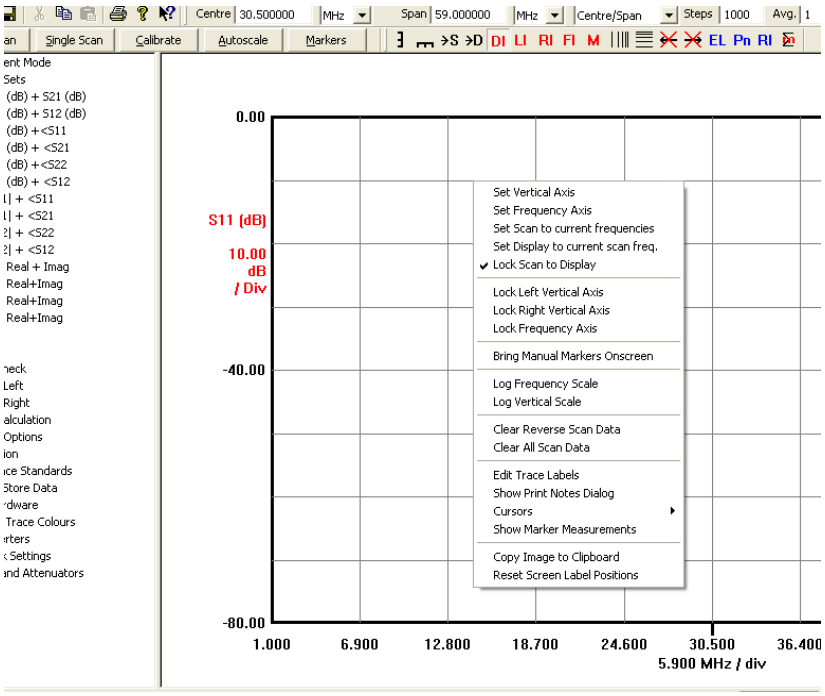
5 Managing the display

This chapter addresses Graphical User Interface subjects, specifically:

- Popup Menus and Toolbars
- Displaying Multiple Traces
- Scan Setting Options and the Scan Toolbar
- Labelling the Screen
- Dragging Items on the Screen
- Locking the Scan and the Display
- On Screen Cursors
- Markers and Marker Arithmetic

5.1 Popup Menus & Toolbars

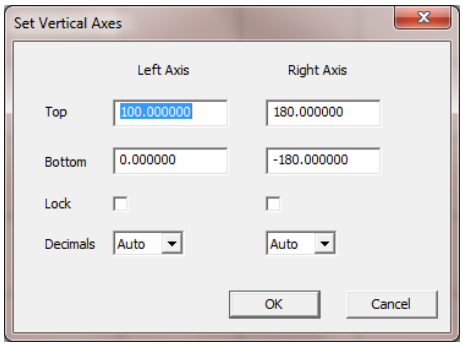
If you right click within the display area you will get a menu like this:

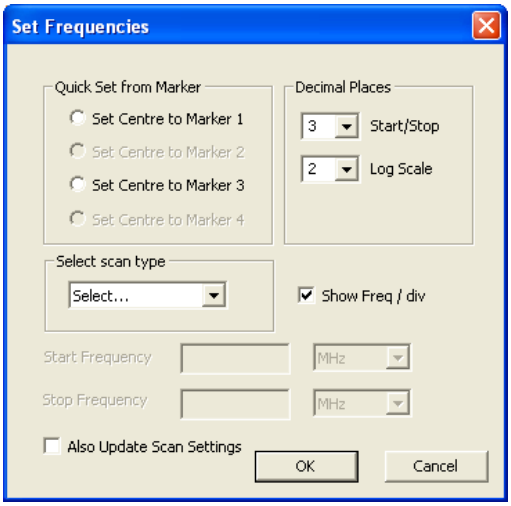


This menu provides some additional control over the display.

Note the two log scale functions. These allow the horizontal or vertical axes to be set to a logarithmic scale. There are some limits here. Obviously, negative values are a bit problematical, so do not expect the display to be correct in all circumstances. Therefore to handle negative values, there is an option under “display options” to allow values to be forced to be positive (displaying |value|) when in logarithmic vertical axis displays. Also note that the snap functions do not operate with a logarithmic vertical scale.

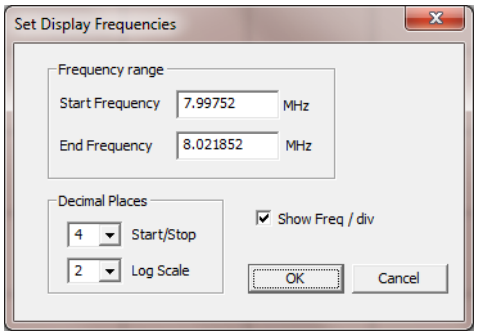
Next, the specific settings of the left and right vertical axes may be set; similarly the frequency axis. Each brings up a menu as shown below





For the vertical axes, the details are self-explanatory. For the frequency axis, you may either explicitly set the frequency using start/stop or centre/span or may set the current centre frequency to the current marker frequency of any currently enabled marker. Note however that the frequencies will only be set if valid for the current calibration data. You may also set the number of decimal places for the frequency axis. The "Start/Stop" settings control the left and right (start and stop) frequencies as well as the centre point in a linear scan. In a log frequency display, the centre value is omitted and the "log scale" setting controls the number of decimal places on each decade point. The "Show Freq/Div" option states the scale on the frequency axis in non-logarithmic rectangular displays.

There are in fact two frequency axis setting dialogs; if the display and scan frequencies are not "locked", then the following version of the "Set Frequency Axis" dialog will be displayed instead:



There is also a second way to access these dialogs. A double click with the left mouse button will cause one or other to appear. If the mouse is pointing to the bottom 10% of the display area or the area under the trace, the frequency dialog will appear, otherwise the vertical axes one will be created.

Returning for the moment to the pop up menu described above, the menu also provides quick access to the axis lock functions.

Locking an axis also protects the axis against changing as a result of autoscale or display type changes. If for example you want to be able to drag a trace up and down

but not to move the frequency axis, locking the axis will allow this. When an axis is locked, a tick mark will appear next to that menu item.

There are three popup menu items that control the linking of display and scan. When linked, the display and scan ranges are the same. When unlinked the display frequency range may differ from the scan. One option links or unlinks them. The other two allow the scan to be set to the current display frequency range or the display frequency range to be set to that of the scan.

Next, if you are using manual markers, it is easier to use this menu to bring then onto the screen than to explicitly set a frequency with the markers dialog. Any manual markers currently set to display will be brought to the centre of the display when the "Bring Manual Markers Onscreen" item is selected.

If you have been rescaling the frequency axis by dragging with the mouse, there is a quick way to set the current main scan to the frequency range you have selected – just click on the "Set Scan to current frequencies" menu item. If the toolbar shows centre/span or start/stop, the frequencies shown will be changed to those currently on the screen. If the current display shows "Full scan" or "From Zero" it will also change to "Start/Stop" first.

Also present in this menu is a shortcut to copy the current display image to the clipboard. Note that you may also use the main menu "copy" command, the copy button on the toolbar or a Control-C shortcut to do the same thing.

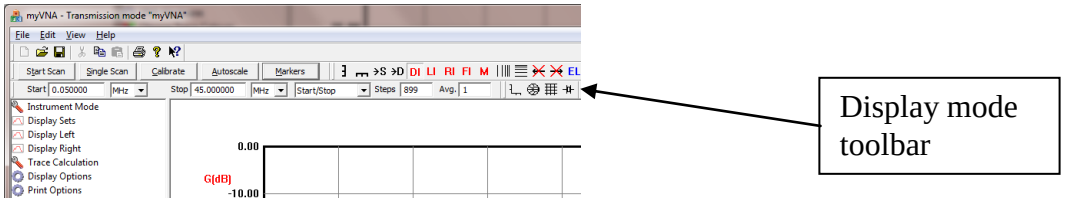
Other menu items open the trace labels and print notes dialogs. Marker measurements dialog may also be opened and cursors placed, edited or hidden.

Finally you may clear the current scan trace data with the "Clear Scan Data" option or just the reverse scan data if using 10 / 12 term S parameter corrections.

There is another way to access common functions, using the "controls" toolbar as shown above. It provides access to the most common functions from the popup menu in the main view.

It is a floating, dockable toolbar.

A second small toolbar permits easy selection of the display mode



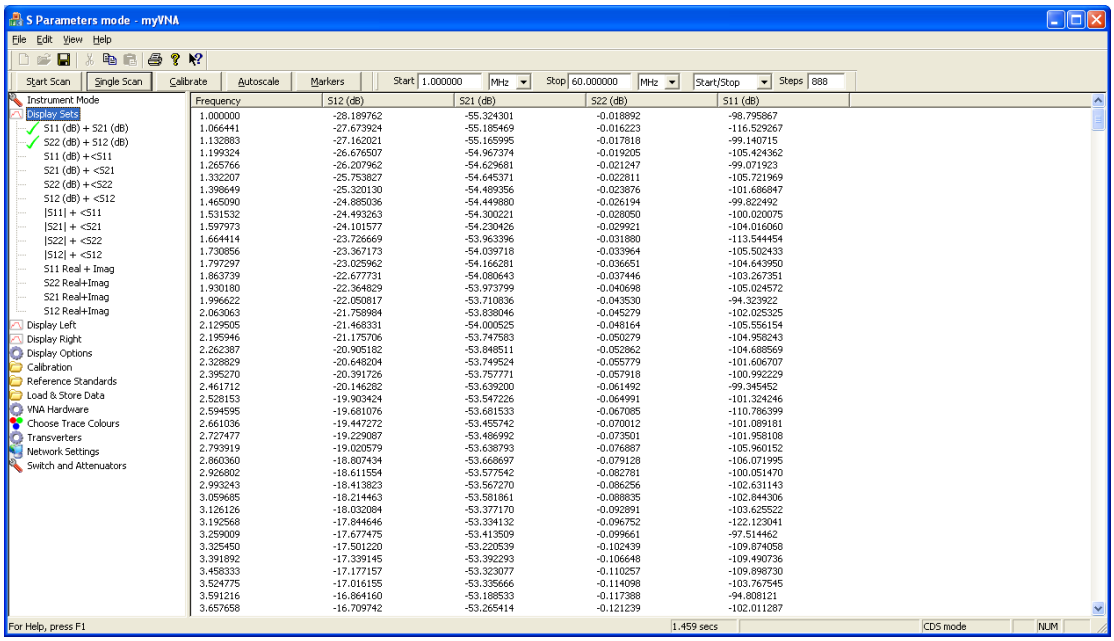
This toolbar, as others is moveable and dockable. Each of the buttons permits easy selection of the rectangular, polar, report and equivalent circuit display modes.

5.2 Displaying Multiple Traces

In all of the above discussion, only one parameter type has been shown on each of the left or right axes by clicking on the desired parameter in a display set, display left or display right option.

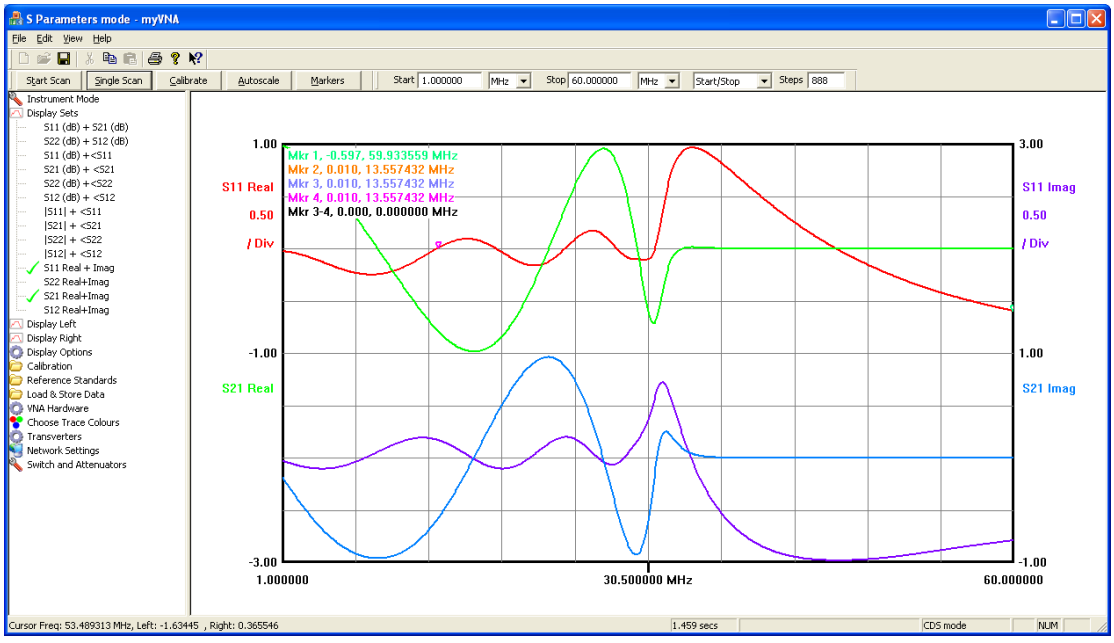
It is however possible to display multiple types at the same time by holding down the 'Ctrl' button whilst selecting a type to display.

In report view this causes extra columns to be displayed, for example as shown below



You may select as many items to display as you want at the same time.

This also works for the main graphical displays, for example:



Again you may have as many as you want at the same time. Note that the trace colours are shown by label on each axis. It can however get quite complex quite quickly.

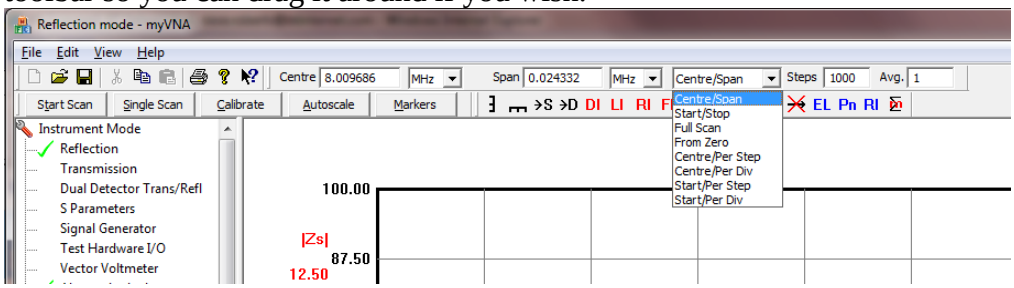
The same principle also applies to data stores. Here however it is not necessary to hold down the control key to make multiple selections; each operates independently.

It is possible therefore to have up to 26 different parameters shown on each axis in a rectangular display and to have the current scan data plus all 4 stores shown at once. You would not want to do this – the display would be unusable – but there is nothing to stop it. The implication is that there would be 260 traces shown on the screen. That needs 260 colours. Clearly impractical.

The current solution to this is not ideal – if I find a better way it will change. There are 8 trace pens available (and visible under the "Choose trace colours" option). The first two are reserved for the "primary" left and right parameters of the current scan data – that is the parameters whose details appear in the upper half of the left and right axes. The other 6 are used for the rest of the traces, cycling round each in turn. In polar or Smith displays only the first pen is "reserved".

5.3 Scan setting options & the Scan Toolbar

Look again at the drop down list of scan settings. The Scan toolbar is the toolbar that extends from "Centre" to "Avg" in the picture below. It is a moveable dockable toolbar so you can drag it around if you wish.



You may set the scan in many different ways. Full Scan sets the range to be the current limits of the VNA. From Zero sets the display from zero to the current "stop" frequency. The other six options are in two groups – those based on a start frequency and those based on a centre frequency. For each, the scan may be set for a specified span, per step or per division. Divisions here refers to the number of divisions on the frequency axis of a rectangular display non log scale. The toolbar will change its labels according to the selection you make.

5.4 Labelling the Screen

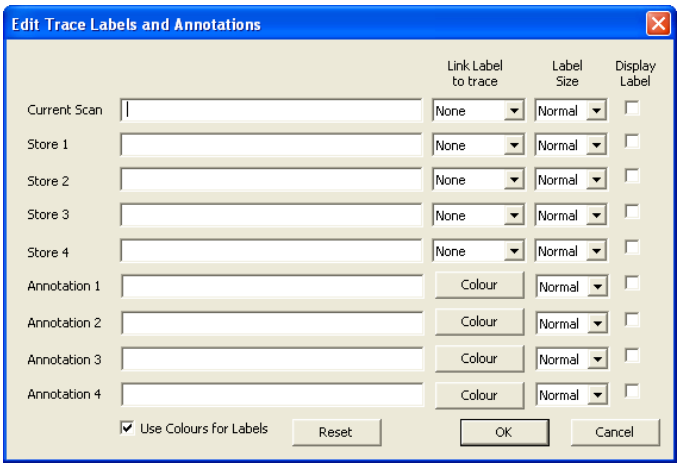
There is now the facility to apply text labels to a display.

There are two types of label – those associated with a trace and free standing annotations. In each case a label may be placed anywhere on the display area and may be moved around the screen by dragging it with the mouse. The difference is that those associated with a trace only display when the corresponding trace is shown and

will alter their position so that their relative location compared to the trace is maintained if the trace moves.

To access the labels dialog, either select “Edit Trace Labels” from the “Display Options” or from the pop up menu after right clicking with the mouse.

Note that this only works in the main rectangular and polar/smith displays and linkage to a trace only works in rectangular displays.



The dialog shown above is used to configure the labels. There is one label for each of the current and saved traces and 4 free standing “annotations”. Each has a string that is displayed and some control over the size of the text (small, normal, large). Each may also be switched on or off with the “Display Label” checkbox.

For the free standing annotations, the colour used may be set; for the trace labels, the colour adopted will be taken from the corresponding trace colour.

Trace labels are linked to a parameter of the trace. This may take one of a number of settings as follows:

- “None” – the label acts like a free standing annotation but only displays when the corresponding trace displays
- “Left” or “Right” – the label is associated with the main left axis or right axis parameter
- Otherwise, the drop down list gives access to all the individual parameters – Rs, Xp, S11 real etc.

Initially the labels will appear in the middle of the screen and may be dragged to where they are needed. For those associated with a trace, if the corresponding trace moves on the screen the location of the left hand end of the label will move to maintain its relative position to the trace as it was when dragged with the mouse; so if for example you place the label just above a point on the trace, it will stay just above the same point should the trace move on the screen.

Labels will display on printout or when copied to the clipboard. They are not preserved when the program exits. Labels may also be set to ignore their associated

trace colours and display in black. Annotations may be set to specific colours by the appropriate buttons on the dialog.

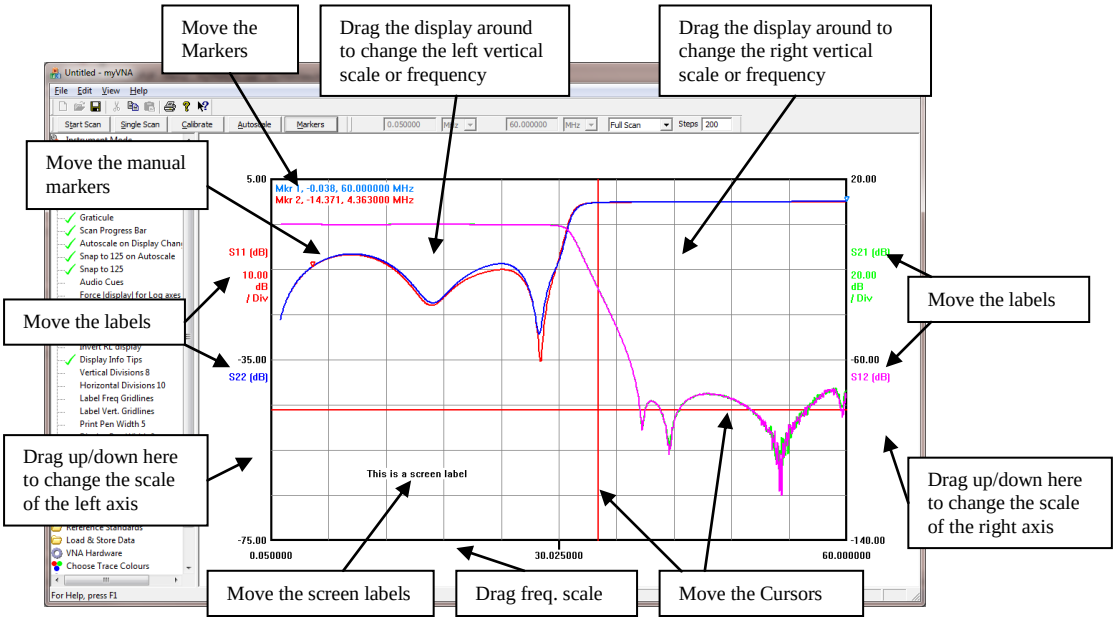
Annotation data (string, position, size and colour) are retained on program exit and reloaded when the program starts again. Note that when this happens, the annotations will display even if when the program last exited, the display flags had been cleared.

When scan data is saved to file in the myVNA native format, any label associated with the data set is also saved into the file and automatically reloaded if the data is read from file.

Labels applied manually to saved data sets are not retained on exit as the data on disk is not rewritten.

5.5 Dragging Items on the Screen

There are various parts of the display that may be moved using the mouse (left click, drag, release).


Move the Markers
Drag the display around to change the left vertical scale or frequency
Drag the display around to change the right vertical scale or frequency
Move the manual markers
Move the labels
Move the labels
Drag up/down here to change the scale of the left axis
Drag up/down here to change the scale of the right axis
Move the screen labels
Drag freq. scale
Move the Cursors
This is a screen label

In addition, clicking on the left or right sides of the main display area other than when over one of the objects shown above will drag the display up, down, left or right, and clicking & dragging to the left, right or bottom of the display permits rescaling the corresponding axis.

To reset moveable text to its default location, right click and at the bottom of the display select the option ‘Reset Screen Label Positions’.

As discussed later in this document, there is a TDR mode. When this is used, the horizontal axis supports both time and frequency. To drag the time axis, hold down the CTRL key whilst dragging as you would for frequency.

In polar view, the markers and screen labels may be dragged around the screen.

5.6 Locking the Scan and the Display

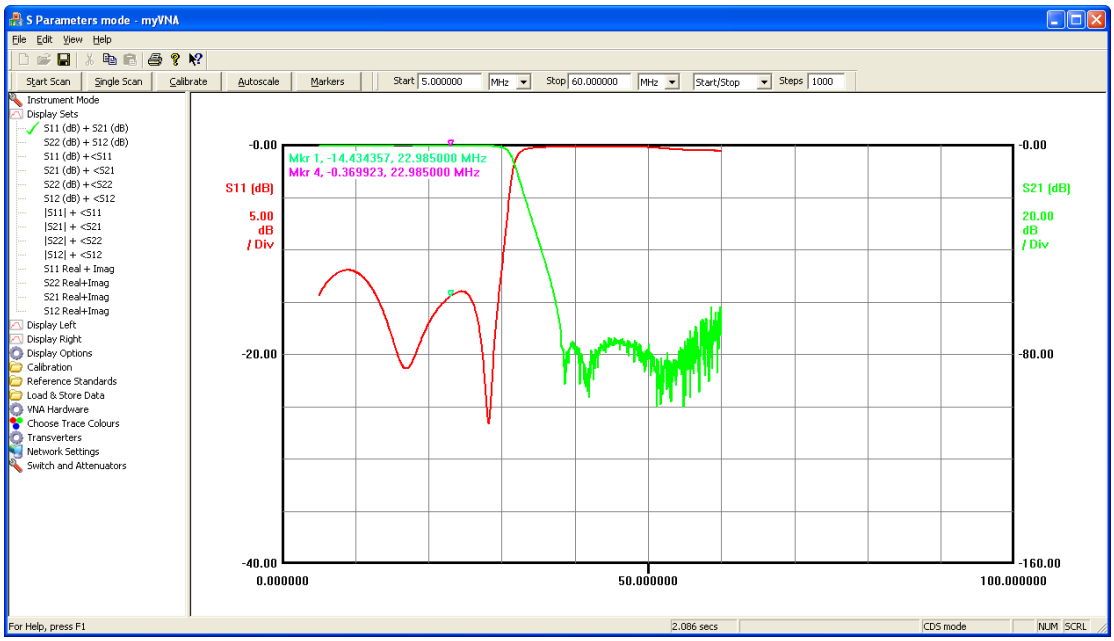
So far, the display and scan have been linked. This means that if you change the scan (start, end, centre, span etc.) the display changes accordingly. Equally if you change the display it changes the scan.

You can also unlink them in rectangular view.

Right click on the display and there is an entry in the popup menu labelled “Lock Scan to Display” that should be ticked at the moment. If this is unticked the display start & end frequencies may be set independently of the scan frequencies. When linked, the limits applied to the display frequency are those of the scan limits. When unlinked they are controlled by a minimum and maximum in the Options under the View menu. The defaults are 0 and 200 MHz.

In this mode the dialog box that pops up when you change the display frequency is replaced by a different one that just sets the display.

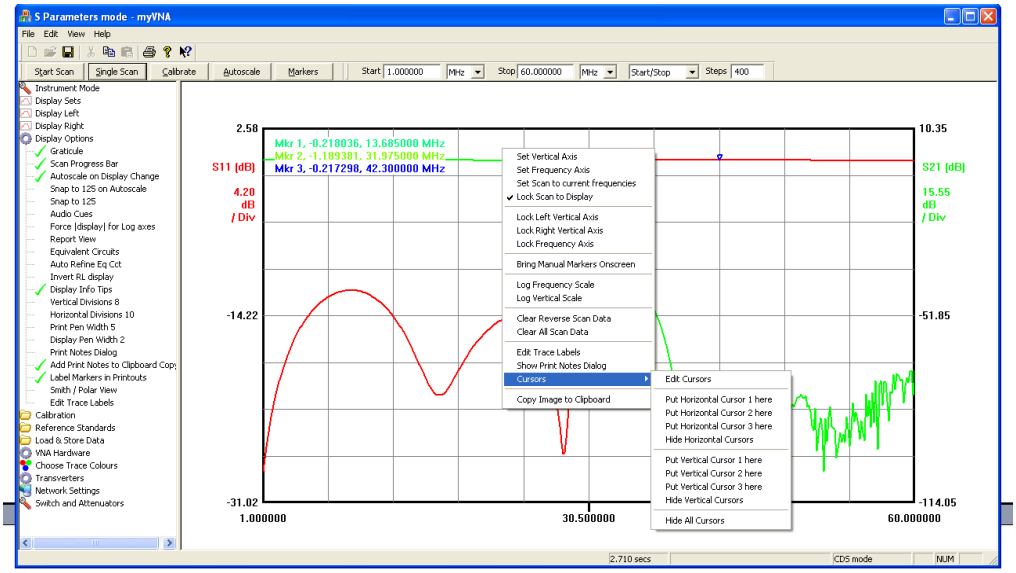
In the example below the scan is set to 5 – 60 MHz, the display is unlinked and the display set to 0 to 100 MHz



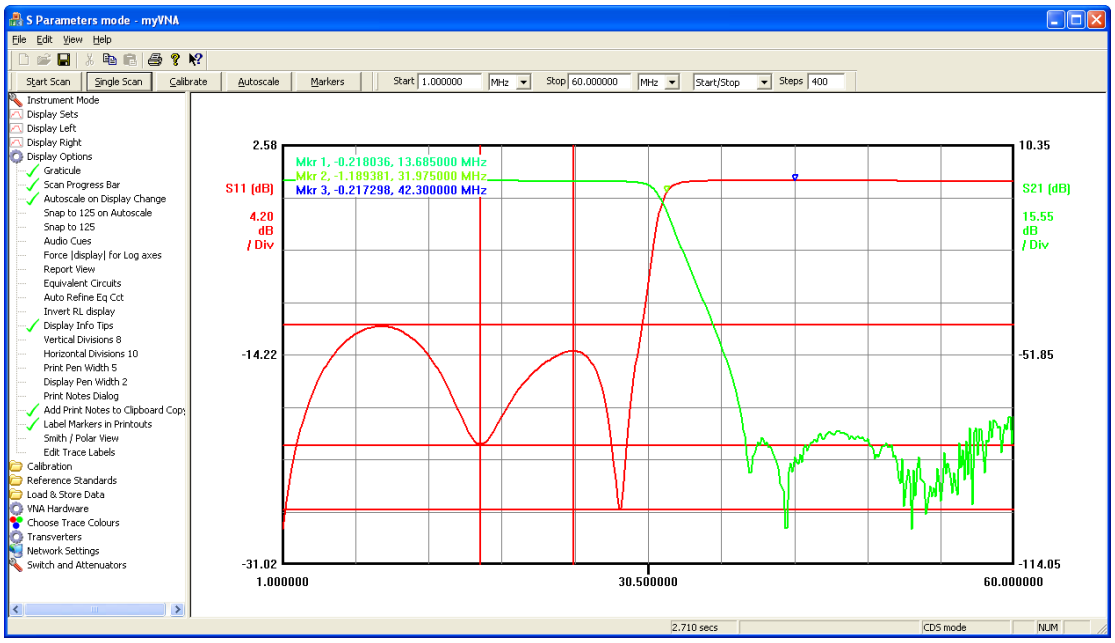
All functions such as markers, log displays, cursor movements should still function correctly.

5.7 On Screen Cursors

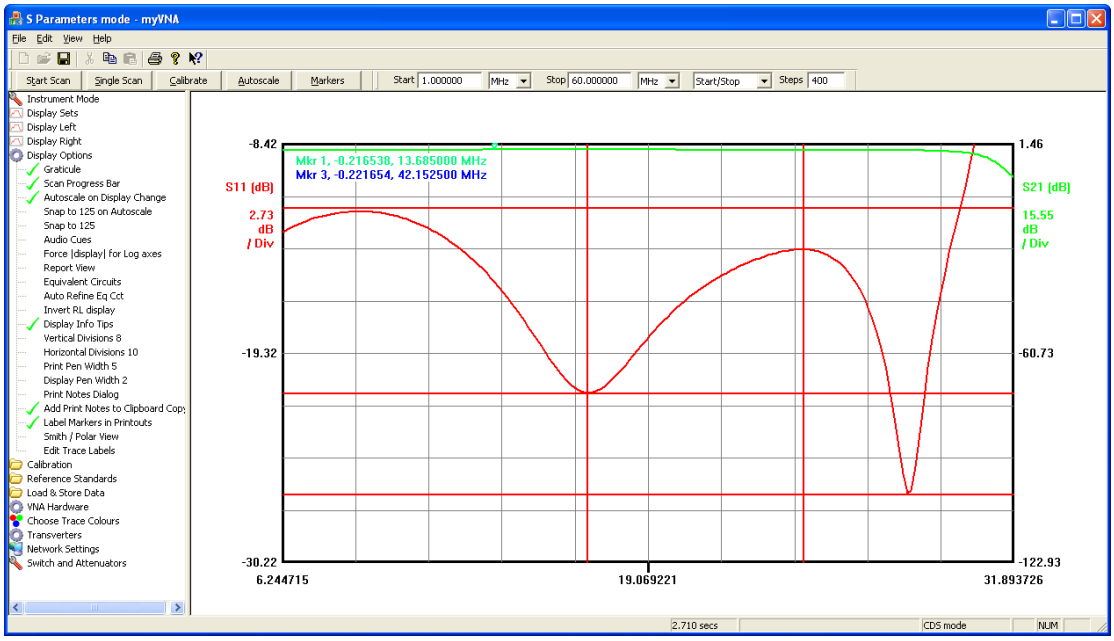
When in a rectangular display, right click to display the menu. One of the options is “Cursors” which brings up a secondary menu as shown below.



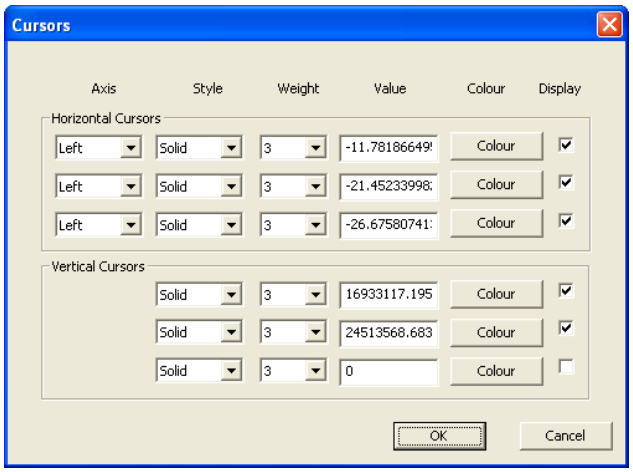
This menu controls three horizontal and three vertical cursors. The simplest and quickest function place a horizontal or a vertical cursor at the position of the mouse when the menu was first opened. The following example shows 3 horizontal and 2 vertical cursors placed in arbitrary locations



As the screen updates, or for example the display is rescaled or dragged, the cursors stay at their positions relative to the scales, so in the next display, the cursors have not been moved but the display has been rescaled by dragging with the mouse



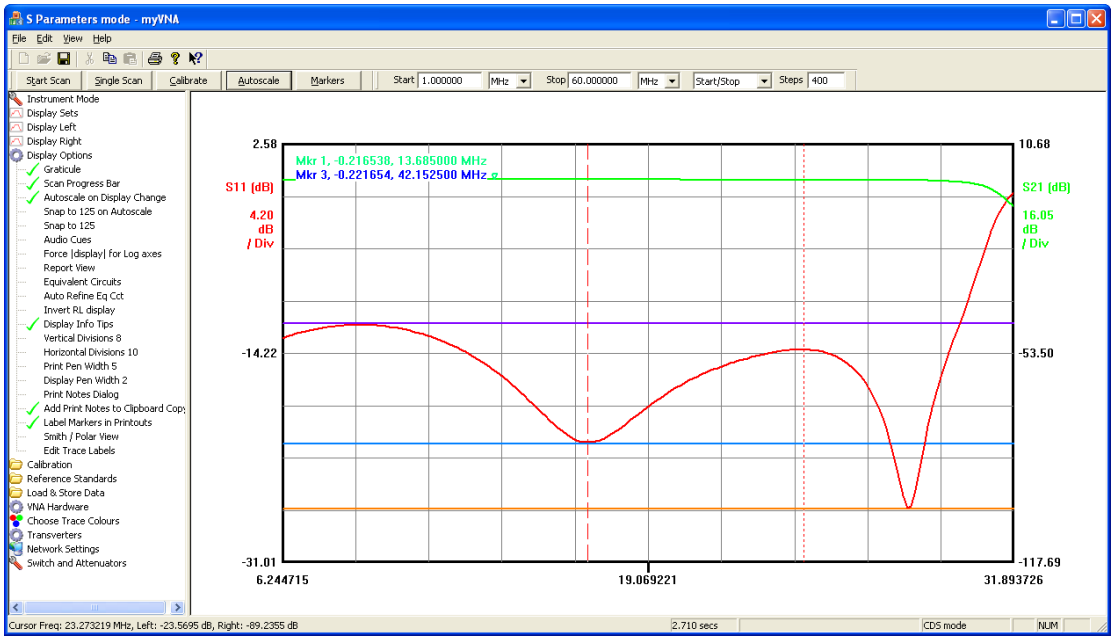
Referring again to the popup menu, another option is to edit the cursors. This produces the following dialog box



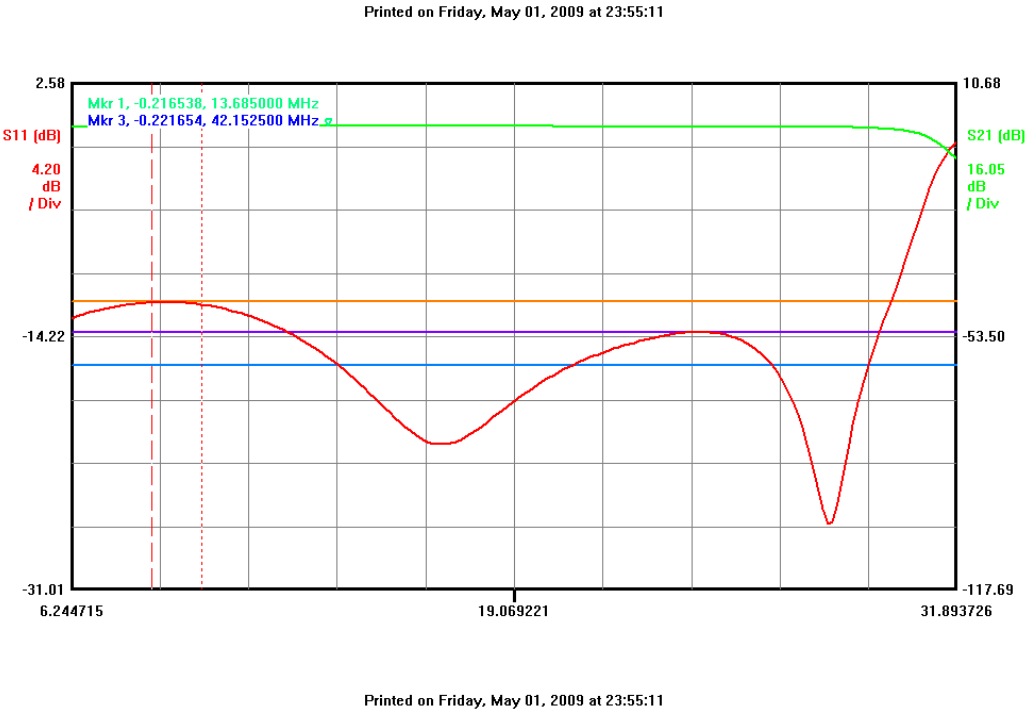
This provides more control over the cursors. Remember above it was noted that cursors stay at their locations relative to the scales? This means that a horizontal cursor needs to know if it sticks to the left or right axis. The type of each may be set – solid, dash, dot or dash-dot line type and the weight (thickness) may be set. Note however that because of the way that the windows functions work, the lines will always be solid for thicker lines.

The value that each cursor is sticking to may be manually set and the colours changed for the lines

Here is the same with line colours changed for the horizontal cursors and line types changed for the vertical ones (note weight was set to ‘1’ for this – see above)



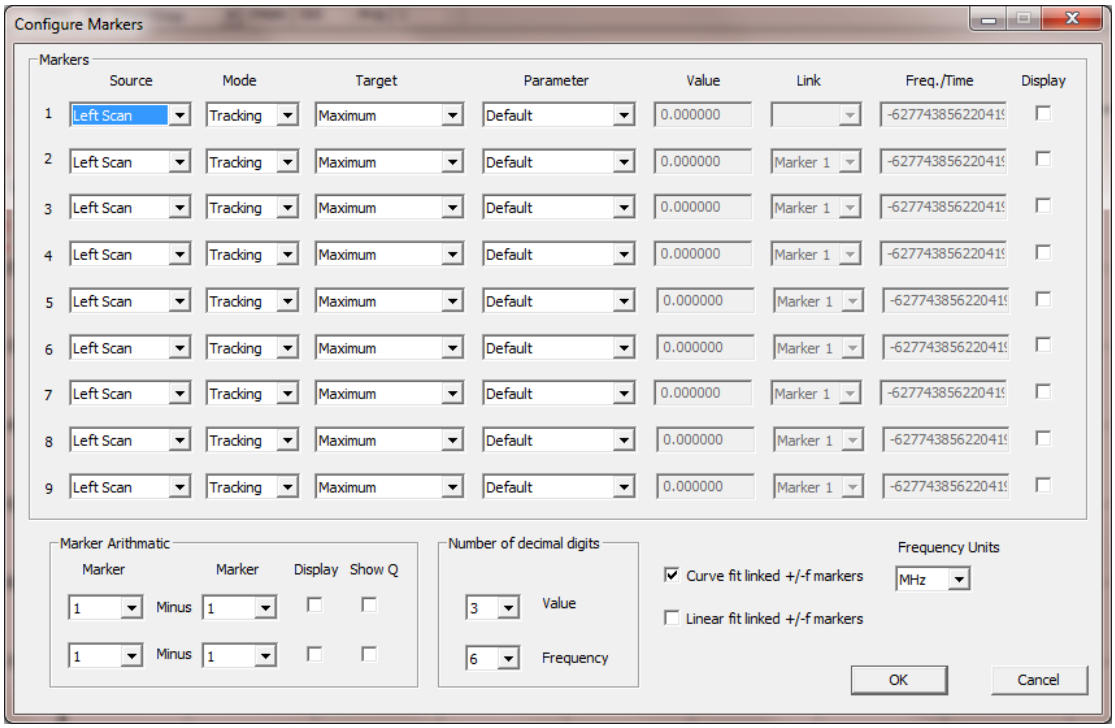
Finally, a cursor may be dragged using the mouse. Here is the result of dragging the cursors with the mouse, and using the clipboard copy function rather than, as in the above traces, a screendump



5.8 Markers and Marker Arithmetic

Markers are used to identify specific points on a trace on the display and to perform calculations on their values.

On clicking the “Markers” button you get a display like this:



There are 9 markers and two marker maths functions.

Each marker may be configured to work in different ways, and pairs of markers may be used together in an arithmetic computation.

Note also that the number of digits to the right of the decimal point may be set for the markers using the drop down list. There are separate settings for the value and the frequency.

Each marker is configured primarily as follows:

- Source
- Mode
- Target
- Parameter

The source defines which trace the marker is attached to; it may be to the current scan data, any of the four data stores or the simulation results, and in addition to a trace attached to the left or the right axis.

The Mode is either tracking, manual or linked. If linked it may also be with an offset to the marker to which it is linked, and this offset may be on the positive (higher frequency) or negative (lower frequency) side.

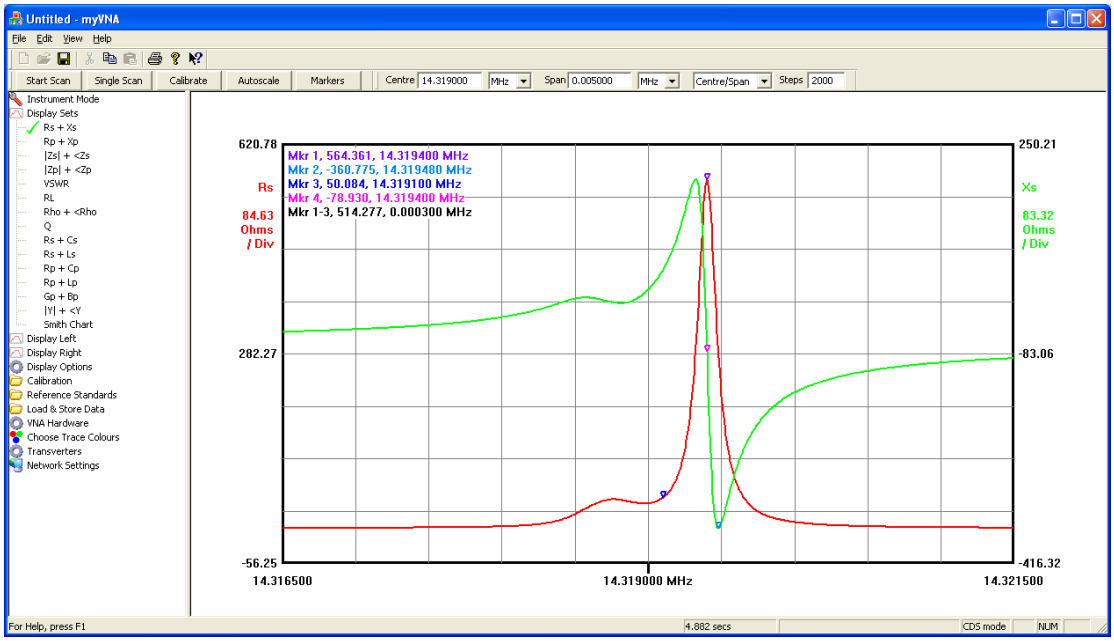
The target setting is used when a marker is set to track, for example if set to track the maximum value, it will move at the end of the scan to position itself at the maximum (peak) value of the trace selected. The target may also be set to track to minimum, or to the first, second or third time a trace crosses a given value, either positive or negative going.

Parameter is used when there are multiple traces shown on a given axis (left or right). Normally it is set to "default" but it may be explicitly set to track any calculated parameter such as Rs, Xs, etc.

Value is used in two different ways. When a marker is set to track a value crossing, this control selects the value to be used. When a marker's mode is set to be linked with an offset (f+ or f-) then the value entered represents the value offset. For example, setting a marker to "f+" with a value of -3, means that the marker will locate a point on the trace on the high frequency side of its linked marker where the value of the parameter is 3 less than the value of the linked marker.

Look to the bottom right; the two checkboxes labelled "curve fit" and "linear fit". Without these ticked markers only locate themselves on the nearest data point in the scan data. With either of these ticked, the value crossing and offset markers will interpolate between scan points to find a more accurate setting. The interpolation may be either a linear interpolation between the points each side or a polynomial between the nearest 3 points.

When a marker is linked to another marker, the "link" control determines to which marker it is linked.



When you perform marker arithmetic, the program will not stop you subtracting, say, degrees from ohms. It is up to you as to whether the units make sense. Also note that if the value of a marker is computed as being over 1e9, the marker will not display.

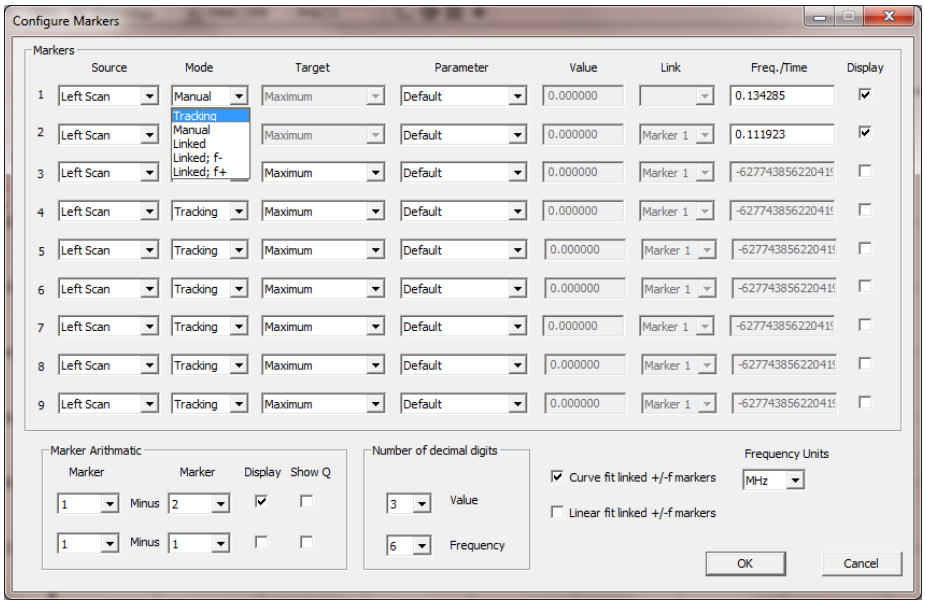
A manual marker will stay put on its frequency, but you can also drag it. If you put the mouse pointer close to it (technically, within 10 pixels) and depress the left mouse button, you can drag it left and right. The marker will stay glued to its trace but can now be moved left & right.

The status bar indicates the current marker position and values as you drag the manual markers. This only lasts whilst the marker is being dragged. When you move the mouse across the screen at other times, the current mouse pointer position will be displayed as frequency & values for left & right axes.

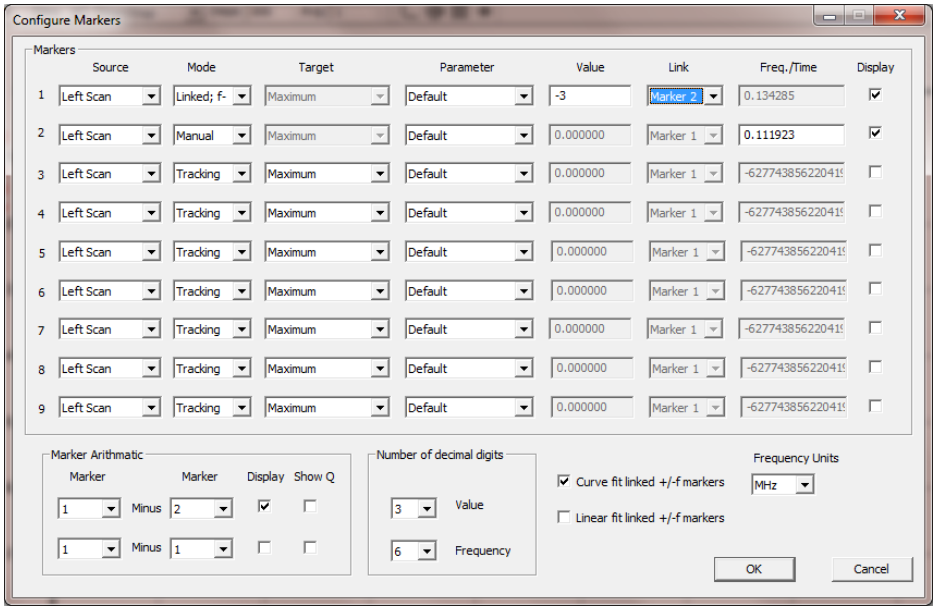
Markers are by default positioned top left of the display. Sometimes this is not very convenient – it obscures parts of the trace. So you may move it around on the screen by dragging it where you want it to be. Anywhere on the marker display text, click the left mouse button and drag the markers where you want them to be. The position you select will be preserved on startup, and there are two positions, one for polar/smith charts and one for rectangular. Finally, there are a few quirks; the position may be off the page on printed copies and dragged off the bottom they may reappear on the top as they strive to stay visible.

In Polar and Smith Chart views, the markers work slightly differently. At their core, they always use the modulus and argument of the measurement. Therefore when setting a marker, the “left” trace refers to the modulus and the “right” trace refers to the argument. Also in Smith Chart view, the values that the markers display may be changed to another parameter, for example Rs+Xs. Underlying this though, the program is still dealing in rho, so the values used in tracking markers will still be rho.

Here is the Marker dialog again. Now look at a couple of other features.

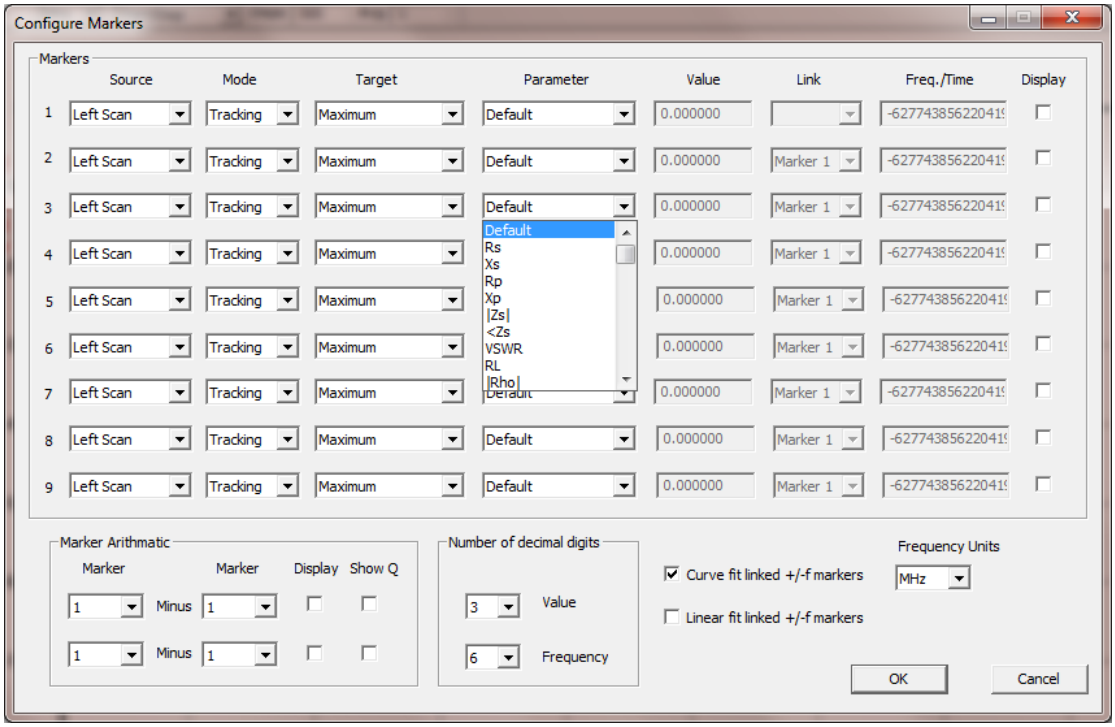


There are two extra options to the mode. Linked with a frequency offset (positive or negative). Selecting one of them enables the value and link boxes thus



In the example above, marker 1 is lined to marker 2 but will seek a value 3 units lower than the marker 2 value. The obvious use is, for example, to position markers 3dB up (or down) from a minimum (or maximum). Setting one each side provides a means to display Q for example. To do this then, link 2 markers, one above and one below, and offset by 3 in a display that is showing dB. Then set the marker arithmetic to subtract the two linked markers from each other. The marker will then display the difference in frequency between the two markers. Finally, tick the "show Q" option. Now the display will be the centre frequency divided by the difference in the frequencies.

Return now to the marker dialog and observe the 'Parameter' column. The default behaviour of each marker is that it is attached to the main (or default) trace. If you display more than one parameter on an axis at a time though, you may want to select the parameter that it is attached to. The Parameter permits this. It opens up a drop down list of all possible parameters. If that parameter is displayed then so will the marker.



5.8.1 Global Show / Hide Markers

On the toolbar is a button like this and correspondingly popup menu items with a right click on the display that permits marker display to be switched on or off.

5.9 Full Screen Display

Under the “view” menu is an option “Full Screen”. If this is selected the display will switch to a full screen display mode. To return to normal, hit the “Esc” key.

5.10 Keyboard Shortcuts

The following keyboard shortcut may be used

Key	Function
Esc	Exit from full screen mode
F1	Open help file
Ctrl-C	Copy to clipboard
Ctrl-O	Load file to store
Ctrl-P	Print
A	Autoscale
C	Calibrate
M	Open marker dialog
T	Start / Stop scanning
S	Single scan
F	Toggle Full Screen
Ctrl-M	Marker Measurement Dialog

6 Save, print and copy

This chapter covers:

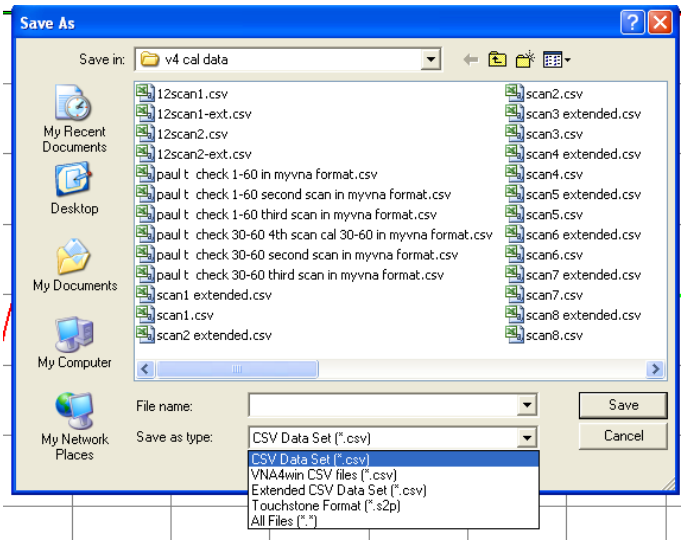
- Saving trace or simulation data to file
- Loading trace data from file
- Autosaving & loading data when exiting ad restarting the program
- Exporting and importing the program configuration
- Printing
- Copying to the clipboard
- Dragging and dropping files from Windows Explorer

6.1 Saving Data to and loading data from Disk

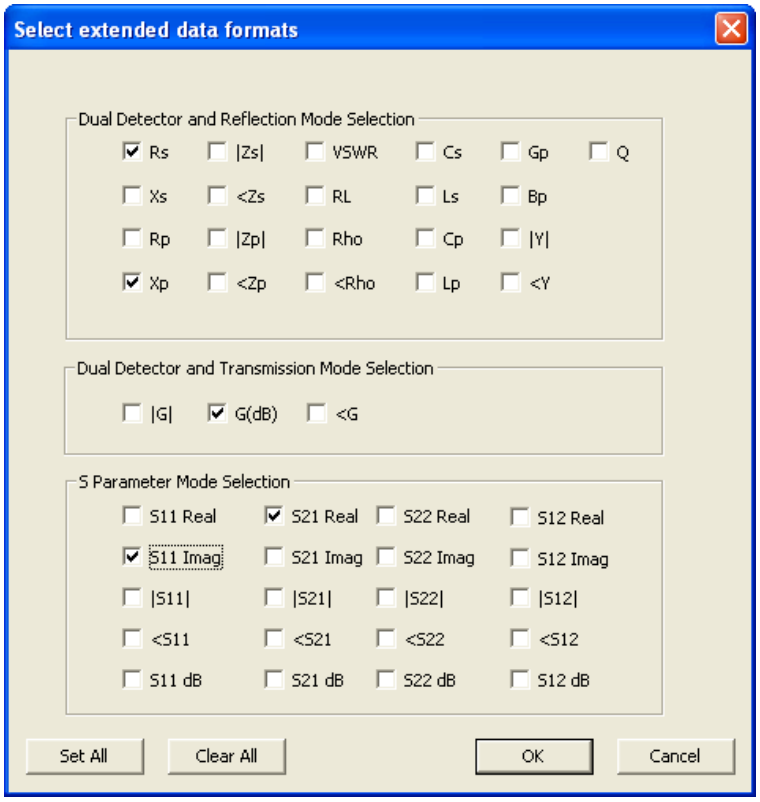
Data may be saved to disk using the "Save to File" options of "Load & Store Data". The data that is saved may be the current scan data, one of the data stores or the current simulation data.

There are four versions of file format supported for saving data and two for loading. The default option saves data in raw measurement form suitable for reloading into myVNA. The VNA4win file format is similar to that used by VNA4win and allows saved data to be imported into Zplots for further analysis. The next format is an extended CSV format and the final one the industry standard "touchstone" file format.

The file format to be used when saving data is selected by the 'Save as Type" drop down list.



Touchstone format is the industry standard SNP format; more on that later. Extended data sets provides the option to select what data types to save. To use it, you must first select what data types to save. This is done using the "Set extended filesave types" option under "Load & Save Data". When selected, it pops up the following dialog box.

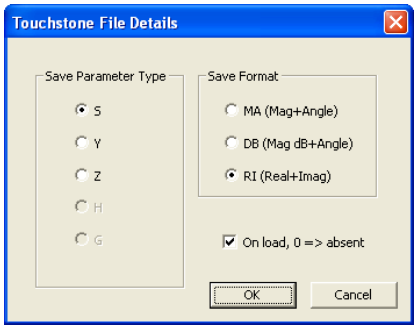

Select extended data formats
Dual Detector and Reflection Mode Selection
Rs |Zs| VSWR Cs Gp Q
Xs <Zs RL Ls Bp
Rp |Zp| Rho Cp |Y|
Xp <Zp <Rho Lp <Y
Dual Detector and Transmission Mode Selection
|G| G(dB) <G
S Parameter Mode Selection
S11 Real S21 Real S22 Real S12 Real
S11 Imag S21 Imag S22 Imag S12 Imag
|S11| |S21| |S22| |S12|
<S11 <S21 <S22 <S12
S11 dB S21 dB S22 dB S12 dB
Set All
Clear All
OK
Cancel

For each instrument mode, you may select what formats of data to be saved when a file is saved in the extended file save form. The settings you choose will be saved on exit and reloaded when the program restarts.

Note that only the default file format or Touchstone – not VNA4win or extended files – are supported for loading back into myVNA.

Touchstone file format (otherwise known as snp file format) is supported for both trace data save and load; but with some caveats. When saving in Reflection mode, data may be saved in s1p format, and when in Transmission or S Parameter instrument mode, s2p mode is available. When loading, formats MA, DB and RI are available in both file types. Units may be in Hz, MHz, kHz or GHz. Files may be saved or loaded in S, Y or Z formats. H and G are not supported.

The program automatically detects the format when loading. To save, a dialog under "Load & Store Data" selects the file details. It is invoked by the option "Set Touchstone File Options" and pops up a dialog as shown below:



Select the parameter type and format. These affect the file when saving. The other option changes the behaviour when the file is loaded. There is a fundamental problem with snp files. They do not cater for saving simple transmission data. The way round this is that the program may be directed to check on loading whether a parameter x11, x12, x21 or x22 where x is S, Y or Z, comprises just zeros. If the tick box shown above is selected, then when the program detects that a data set has parameters of all zeros it will interpret that as being missing data as opposed to data that is genuinely set to zero. In most cases the setting to use is to have this box ticked. One final caveat – the program ignores the R parameter on file import. Incidentally, if the Options line is missing it follows the Touchstone 1.1 convention of defaults – GHz, S and MA.

6.1.1 Autosave

There is an option under "Load & Store Data" titled "Auto Load/Save trace data" . When selected, the program will save to disk the contents of any of the 4 data stores that are showing data when the program exits and, if selected, will reload those data stores when the program restarts.
The data are stored to the current user specific Roaming Appdata directory under the subdirectory myVNA.

6.1.2 Load and Store options and configuration

The options provided under the "Load and Store Data" treeview control for load & store are as follows:

"Save Scan to store x"	the current scan data is copies into a store (1 to 4)
"Save simulation to store"	the current simulation results are placed in a store
"Save xxx to file"	the current scan simulation or contents of a store are saved to disk. The save may be in the program internal format, VNA4win, extended csv or Touchstone snp format.
"Load file to store"	a csv or snp file is loaded into one of the stores
"display store x"	toggle on or off the display of a store
"set extended filesave types"	controls which data are written to an extended csv file
"Set touchstone file options"	determines the snp file options
"Set scan to file on load"	if selected, then whenever a file is loaded to store, the scan (frequency & number of points) is set to match the loaded file
"Set scan to stored trace..."	permits the scan to be set to match that of a current store

6.1.3 Export & Import Program Configuration

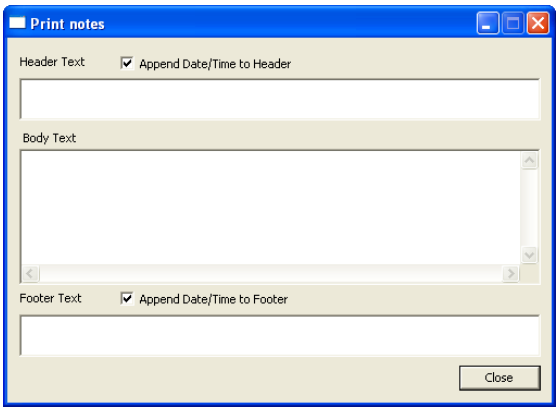
Two options under "Instrument Mode" permit the export to a text file and import from a text file of the current program configuration. This is separate to the main system configuration load & save to the registry that still takes place on program start / exit. Whilst the file is structured in the style of an "ini" file, it is not recommended that you edit this file manually. You may do so - the program will not try to stop you – but the results may not be as intended!

6.2 Printing

You have the option either to print directly or to copy an image to the clipboard that may be pasted into another document. Both functions will preserve the aspect ratio of the screen display currently selected in the printed or copied image.

When printing a trace, you may now add notes to the printout. Under “Print Options” there is an entry labelled “Print Notes Dialog”. It is also available on the right click popup menu. This dialog is as shown below. It allows control over the header, footer and notes sections of the printed page. The Header and Footer are two lines long each that appear centred on the page, top and bottom. Each may also be extended to a third line by including the print date & time using the appropriate tick box. The notes section is a freeform text area that appears beneath the trace display. It will be truncated at the end of the page if it is too long. Noting that the display may fill the page length, especially in portrait prints, some space needs to be made for the notes. When necessary therefore the program will shrink the plot area to leave space, but will only shrink it sufficiently to allow up to 1/3 of the page height to be taken up by the notes. After that the notes are truncated.

The notes dialog may be left open on the screen when the program is in use, or it may be closed and reopened. When this happens the previously entered text will be remembered. The Header & Footer are preserved on program exit however the body text notes are cleared on program startup.



The Notes dialog box may also be resized to suit your needs

If you want to change the width of the traces shown either on screen or when printing, there are separate settings for each on the “display options”. The values represent the pen width in pixels. The graticule will always be 1 pixel thick and the border around the display will be 2 or 3 wider than the trace pens. The size of the markers will change to reflect the resolution of the display.

Also available are two options to change the fonts on a printout. The notes font applies to the header, footer, print notes and (if present) the marker calculations. The other font applies to the fonts on the trace graphic.

6.3 Copy to Clipboard

On the main toolbar is a "copy" button that copies current data to the chipboard. If the current display is the main trace, either a polar or rectangular display, the current trace data and labels will be copied as an image to the clipboard. It may then be pasted into another program such as Word, Excel, Paint etc.

There is also a tickable option under "Display Options" labelled "Add Print Notes to Clipboard Copy". When this is ticked, the header, footer and notes from the Print Notes dialog will be added to the image copied to the clipboard. This function does not apply to the equivalent circuits display (yet).

If the type is "Equivalent Circuit" the controls shown on the screen are omitted from both the printed or clipboard images and the schematic is displayed in their place.

If however the current display is the report view, then the data is placed on the clipboard in both CSV and Text formats. This means that you may paste directly into, for example, excel. If you do this as a simple paste though, it will be pasted as text with each row of comma separated data appearing as text in a single cell – not very useful. Excel though has two paste functions – Paste and Paste Special. If you use paste special, either from the Excel Edit menu or by right clicking in the display and selecting 'Paste Special' you will also be presented with an option to paste CSV data. Do this and it will be pasted correctly.

Within myVNA in Report view, when you click the 'copy' button, two different types of copy may occur. If no rows of the display are selected, all current data is placed on the clipboard. If however one or more rows are selected (for example by clicking on one or more of the rows in the frequency column) then only those selected rows are copied to the clipboard. You may select a block of data by clicking on the first row, holding Shift down and selected another row, or you may select individual rows by holding down the Control key and clicking on specific rows.

When Report View is selected, it is also possible to copy just one column of data (or a subset of a column by using the mouse with the keyboard Shift and Control buttons). To do this, right click on the display area and a popup menu will appear. For whatever columns are currently displayed, a menu item will be present to allow that column's data to be copied to the clipboard. It will also be in both text and CSV formats, but in this case as there is just one column, there will be no trailing commas. This format is suitable for pasting data into Lotus 123 as I can't figure out how to make it paste properly yet.

Finally, it is possible to change the column order by dragging columns left or right using the column heading row. When this is done, the order in which the columns are copied to the clipboard will match the currently selected order. Beware though – this order will be reset every time 'report view' is selected.

Marker measurements data may also be copied to the clipboard with the copy button in that dialog box. This will copy all current rows or (if one or more rows are selected) just those rows. As for the report view, the data is available on the chipboard in both text and CSV format to permit pasting into excel or word.

6.4 Drag & Drop

If a file is dragged onto the application display window using windows Drag and Drop, the program will attempt to load it. It will use the file extension to try to determine what the file is. If it is a .ref, it will try to load it as new reference standard offset data. If it is a .cal file it will attempt to load it as new calibration data.

If it is .ref, .csv, .s1p or .s2p it will attempt to load the data into a saved trace store. If one file is dragged, it will be loaded into store 1. If a set of files are dragged in one operation they will be loaded into stores sequentially until all stores have been loaded; any remaining files will be ignored.

7 Program Limits and other options

This chapter defines some of the program limits and options and how these may be changed.

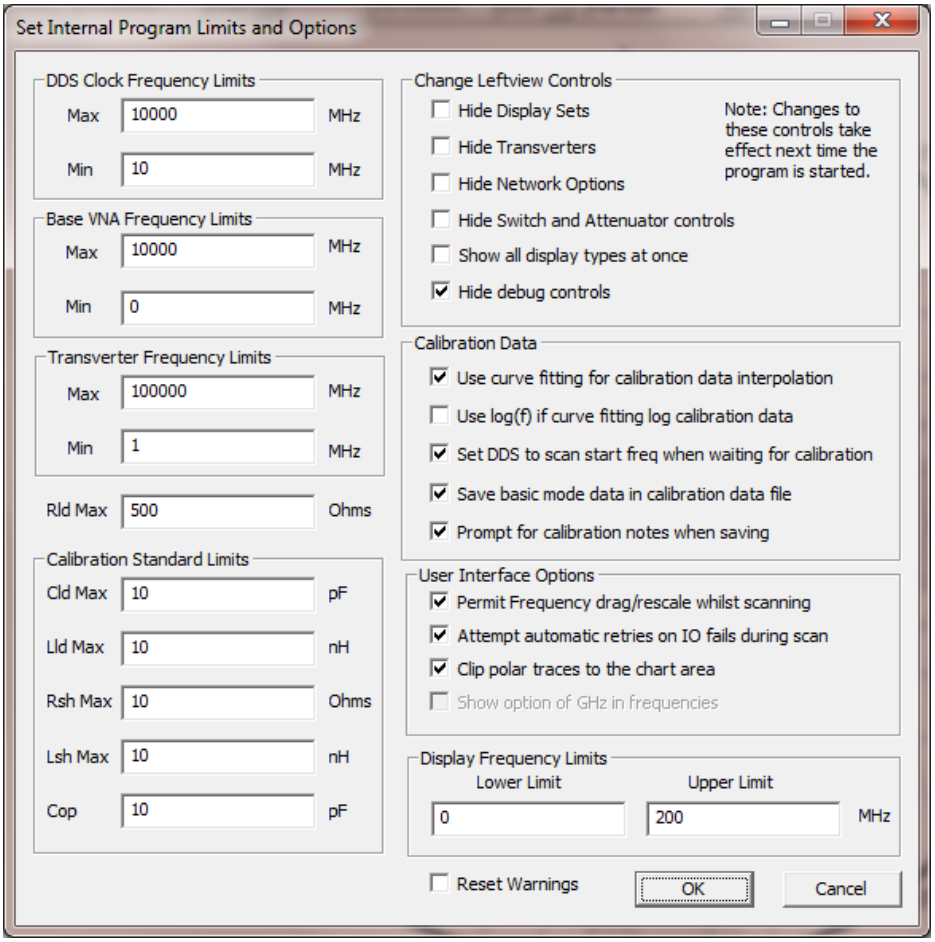
The number of scan points can be changed with the ‘steps’ setting of the frequency bar. The default number of points is 200, but it can be set anywhere from 20 to 50,000. Note that if you go over 2000 points the program will increase buffer sizes and consume more memory. The number of scan points is remembered when exiting, but if it exceeds 2000, then a value of 2000 will be used next time the program starts.

Averaging allows up to 10,000 averages per reading.

Delays between steps can be set up to 200,000 µsec.

A fairly simple bit of maths will show that if you set the program to 50,000 steps with 0.2 seconds between every DDS change and average 10,000 points per step in 32 point harmonic suppression mode you will be there for an unfeasibly long time (approx. 102 years to be precise).

The program has a number of built in limits. In general you should not need to change any of these, but if you do, you will find them in a dialog box accessed from the main menu, view, options. It pops up a dialog box as shown below. These configure the various internal program limits and also permit the hiding or display of optional features should you not want to use them. Some limits, like the maximum number of steps and averages are not changeable.


Set Internal Program Limits and Options
DDS Clock Frequency Limits
Max 10000 MHz
Min 10 MHz
Base VNA Frequency Limits
Max 10000 MHz
Min 0 MHz
Transverter Frequency Limits
Max 100000 MHz
Min 1 MHz
Rld Max 500 Ohms
Calibration Standard Limits
Cld Max 10 pF
Lld Max 10 nH
Rsh Max 10 Ohms
Lsh Max 10 nH
Cop 10 pF
Change Leftview Controls
Hide Display Sets
Hide Transverters
Hide Network Options
Hide Switch and Attenuator controls
Show all display types at once
Hide debug controls
Note: Changes to these controls take effect next time the program is started.
Calibration Data
Use curve fitting for calibration data interpolation
Use log(f) if curve fitting log calibration data
Set DDS to scan start freq when waiting for calibration
Save basic mode data in calibration data file
Prompt for calibration notes when saving
User Interface Options
Permit Frequency drag/rescale whilst scanning
Attempt automatic retries on IO fails during scan
Clip polar traces to the chart area
Show option of GHz in frequencies
Display Frequency Limits
Lower Limit
Upper Limit
0
200
MHz
Reset Warnings
OK
Cancel

If you change the configuration of which options are shown in the left treeview, note that the changes will not take effect until you restart the program.

There are two options that alter the way calibration data is generated. When you perform a calibration, it is performed with a specific number of points. When you subsequently use that data, you may choose a different number of points, a narrower frequency range or a logarithmic frequency scan. In all such cases the program needs to generate a scan specific set of calibration data. When it does this it may need to interpolate between calibration data points. With the "Use Curve Fitting for calibration data interpolation" box unchecked, then a simple linear interpolation between points will be used. However, a better fit may well be obtained using a curve fitting algorithm. With the second box ticked it will do just that. The second tick box comes into play when a log frequency scale was used for the calibration data. In this case the curve fitting is modified to take this into account also. The effect of this appears to be small so the default configuration is to tick the upper box and not the lower one but feedback on their effect would be appreciated.

When a frequency scan is in progress, dragging the scale left or right or altering the scale can be confusing on anything but a fast PC. Therefore you may enable or disable this feature by ticking the "Permit frequency rescale/drag whilst scanning" box.

The option to automatically retry on an IO failure changes the behaviour of the program when a scan completes. When a scan completes, a check is made as to whether it has been as a result of an error such as an ADC timeout. Normally, a dialog

box will be displayed indicating the reason (if one exists). When this option is selected, this will not happen. Instead, the program will try to restart the scan. If the error is a persistent one, such as VNA power failure, the result will be an error, but if it was a transient such as an ADC timeout or overrange, the scan restarts (or tries to).

8 Smith and Polar Charts

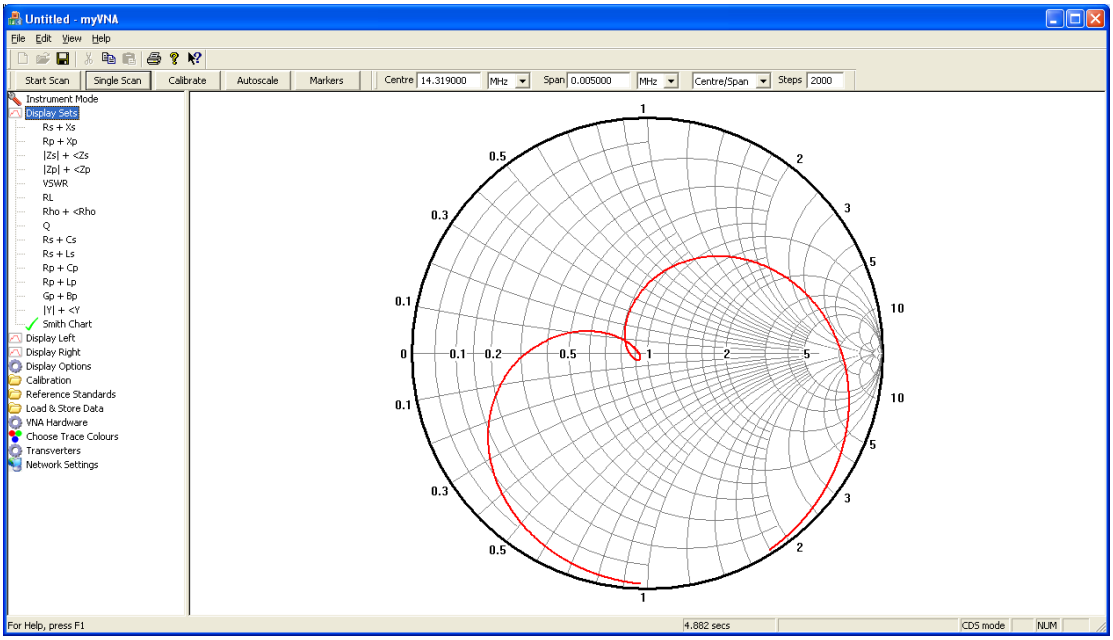
Data may be displayed in four formats:

- Rectangular plots (all traces so far in this manual)
- Smith and Polar charts
- Report (Listing) view
- Equivalent circuit view.

This chapter covers Smith and Polar views.

Under “Display Options” is the option “Smith/Polar view”. When selected, a polar or Smith chart is displayed instead of a rectangular display. In all polar modes, the “Display Left” and “Display Right” lists are empty; the display options are shown under “Display Sets” for all polar modes. When the selected mode is a reflection, S11 or S22 parameter, a Smith chart is displayed. In transmission modes, a polar display is shown. The polar display may be either a log or a linear radial scale. Clockwise around the display is the angle in degrees.

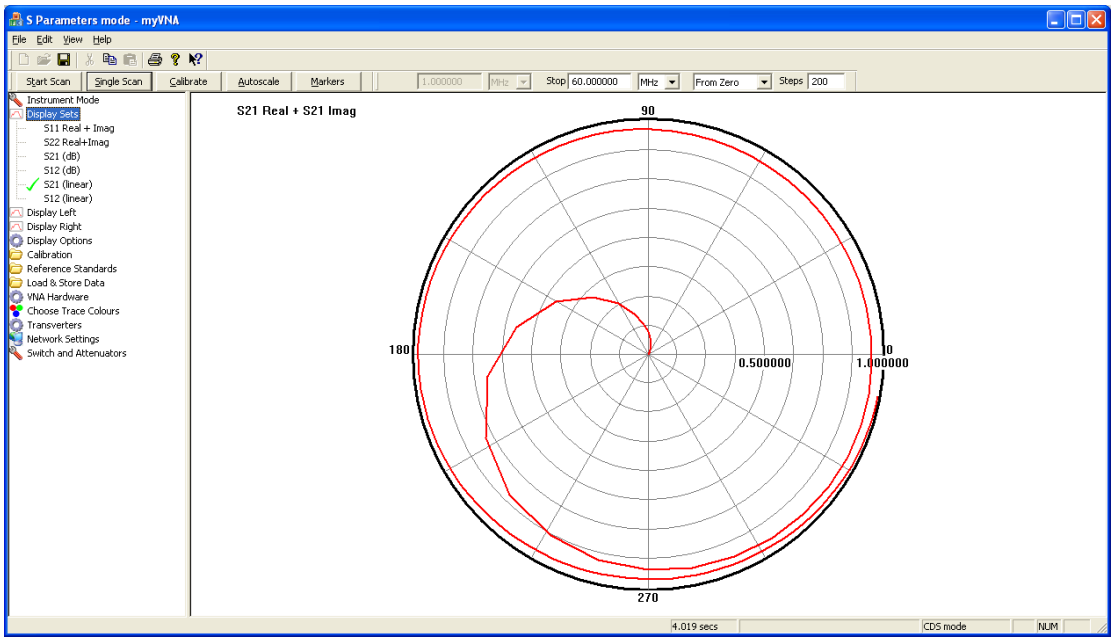
Limited rescaling is available, and only in the polar modes.



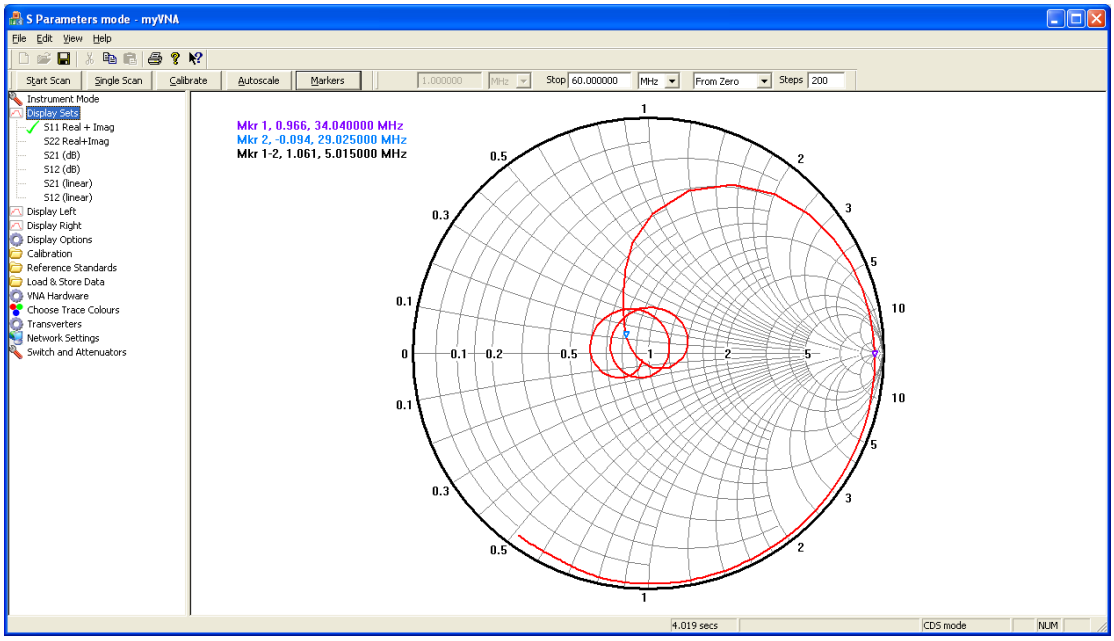
Markers also function in Smith Chart mode. When switched to Smith Chart, “left” trace refers to modulus of the signal and “right” refers to phase. Functions such as seeking for a crossing point, max or min will function but note that the values used are |Rho| and <Rho not normalised impedances. Compared to the rectangular plots of the other options, dragging in polar mode is different. Position the mouse near the manual cursor (don’t forget, right click, bring manual cursors on screen will make them appear) and press the left mouse button. As you drag *left* and *right*, the cursor will move in *polar* coordinates around the trace. Note also that the marker may jump when you start to drag; this is so that it remains possible to position the cursor to any frequency on the scan without having to release the mouse button and drag again.

In polar display modes, markers will display as two values followed by the frequency. The first value is the absolute value of the parameter, for example |gain| or |rho|. The second parameter is the phase angle in degrees. In Smith chart view, this may be further refined via the popup menu to change the display for each marker. A choice of values such as rho, rs+xs, rs+cs etc. is provided. These options are not saved on exit.

Finally the program permits some display scaling, for example when displaying in log display modes. However the display will get quite confused if you display smith, linear polar and log polar at the same time. By all means do it but don't expect it to be useful.



By default, the trace will be clipped to the edge of the chart. If for some reason the trace extends beyond the outer edge, it will appear during the scan (as a black line) but when redrawn at the end of the scan it will be clipped. If you want to see the trace beyond the edge, then you can untick the option "clip polar traces to the chart area" in the program options.



8.1 Zooming

There is the facility to zoom in to polar and smith charts.

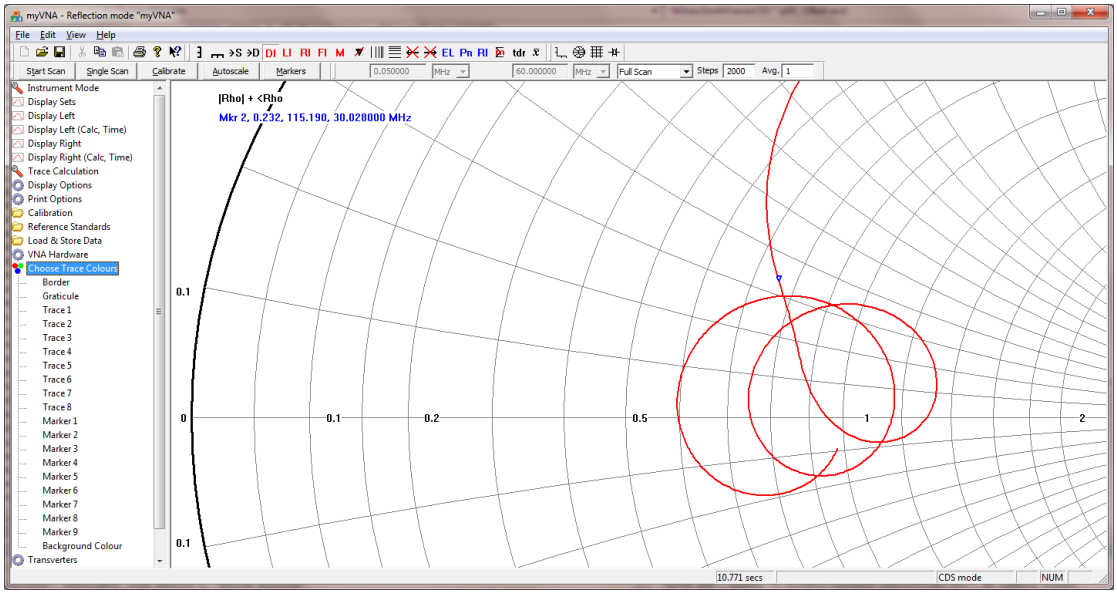
There are two ways to zoom in and out. Firstly, from the pop up menu there are three options:

1. Zoom in
2. Zoom out
3. Reset zoom view

Secondly, two hotkeys may be used to zoom in and out. Hold the control key down and depress the '+' or '-' keys.

You may zoom in by a ratio of up to 10x.

It is also possible to drag the displayed area around to zoom in on a specific area. The display below shows the effect of zooming in and dragging the display to show a specific region. To drag the display, press the left mouse button, move the display then release the button.


myVNA - Reflection mode "myVNA"
File Edit View Help
Start Scan
Single Scan
Calibrate
Autoscale
Markers
Full Scan
Steps 2000
Avg. 1
Instrument Mode
Display Sets
Display Left
Display Left (Calc, Time)
Display Right
Display Right (Calc, Time)
Trace Calculation
Display Options
Print Options
Calibration
Reference Standards
Load & Store Data
VNA Hardware
Choose Trace Colours
Border
Graticule
Trace 1
Trace 2
Trace 3
Trace 4
Trace 5
Trace 6
Trace 7
Trace 8
Marker 1
Marker 2
Marker 3
Marker 4
Marker 5
Marker 6
Marker 7
Marker 8
Marker 9
Background Colour
Transverters
|Rho| ∠Rho
Mkr 2, 0.232, 115.190, 30.028000 MHz
0.1
0
0.1
0.2
0.5
1
2
0.1
10.771 secs
CDS mode
NUM

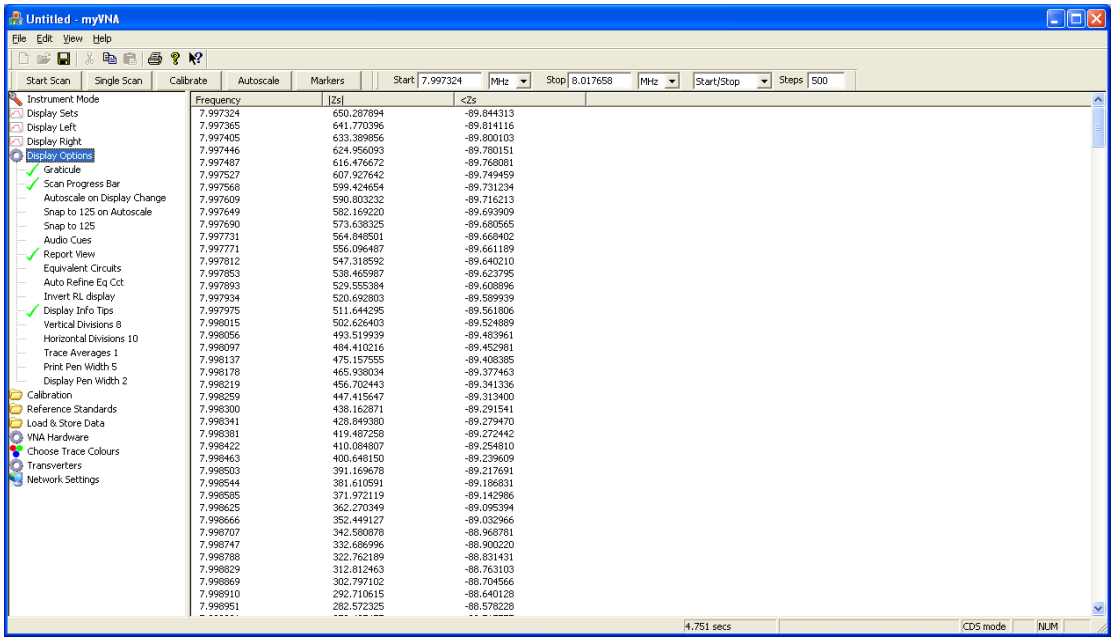
9 Report View

Data may be displayed in four formats:

- Rectangular plots (all traces so far in this manual)
- Smith and Polar charts
- Report (Listing) view
- Equivalent circuit view.

This chapter covers Report view.

Under “Display Options” there is an option of “Report View”. If selected, this changes the graphical display to a tabular report as shown below.



The columns will be whatever the current displayed parameters are (and you may change them in the view) and the column widths may be adjusted by dragging the divider lines between the headings. The order of columns may also be changed by dragging the column header left or right. A scroll bar will appear on the right to allow the data to be viewed, and if printing is selected from this view, a tabular report will be generated. Copying to the clipboard is supported. If one or more rows are highlighted (click on the frequency) then only the rows selected will be copied to the clipboard. If none are selected the whole data set is available in text form on the clipboard. Via a right click popup menu, it is possible also to specify a single column (or selection from a column).

If time is displayed in report view, the time data will be displayed as requested but not the time axis. This is because the program would need to display separate axes for each of the available data source time windows making the data confusing. If you use a report view TDR display, the axis for each value will be the time window for the corresponding data source (not the display window).

One limitation – it will get confused if you have a store and a scan displayed in the report view at the same time with different limits (frequencies, points, log/lin).

10 Equivalent Circuits

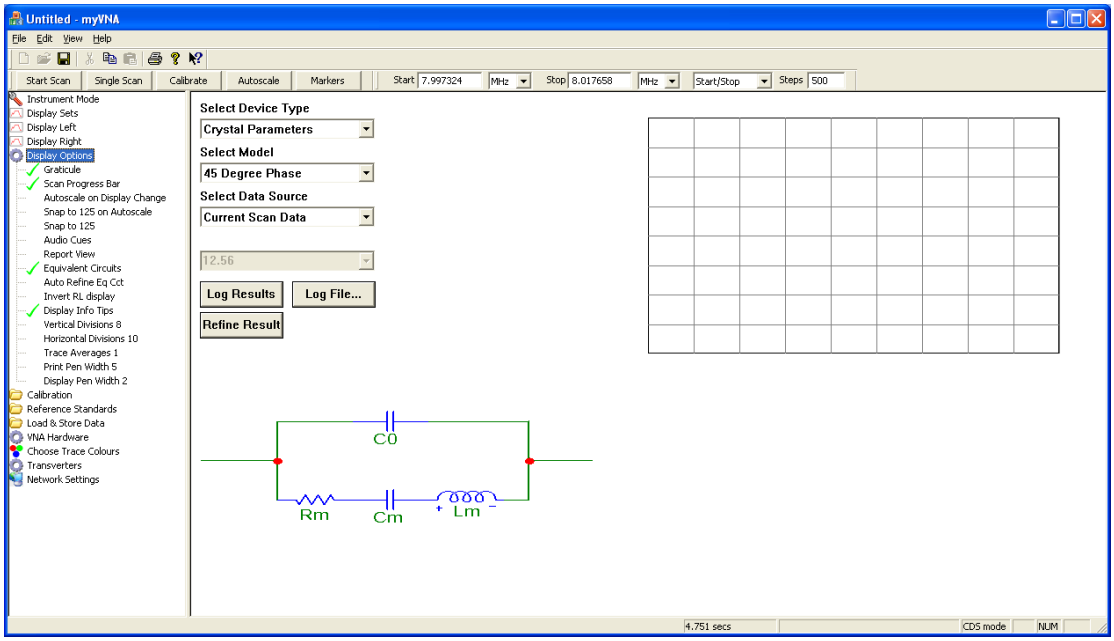
Data may be displayed in four formats:

- Rectangular plots (all traces so far in this manual)
- Smith and Polar charts
- Report (Listing) view
- Equivalent circuit view.

This chapter covers the Equivalent Circuit view.

There is an “Equivalent Circuits” function on the “Display Options” screen. It works in a similar manner to the tabular view; you may switch between it and the other views.

Select it and the display changes to something like this:



What is displayed is governed by the Instrument Mode and the drop down list settings on this screen. When you select “equivalent circuits”, the program will change the display types to suit it; you can change them whilst in this screen but the results may be wrong.

It works in Transmission mode or Reflection mode but the only device type for now is “Crystal Motional Parameters”. For each device type a number of models may be available and the data source may either be a stored data set or the current scan data.

The display has 4 parts to it. Top left, the program is configured by means of the controls. Top right, there is a small display of the trace data as would be seen in the normal view (but simplified). Bottom left is the equivalent circuit. Bottom right is where the results will be shown.

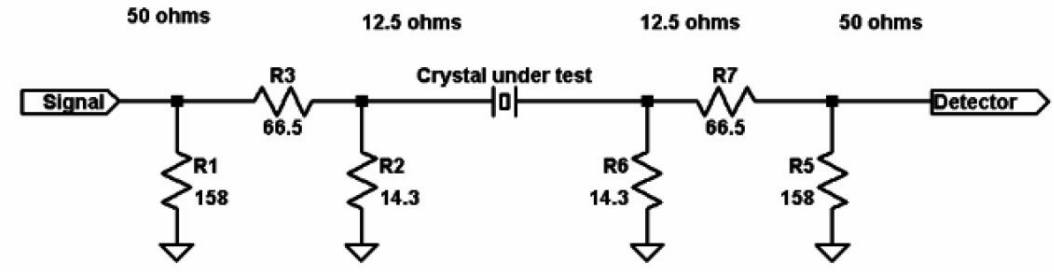
After a successful measurement, the results will be displayed bottom right and may be logged to a file.

10.1 Crystal Motional Parameters

The model used is taken from an excellent paper by Jack R. Smith K8ZOA "Crystal Motional Parameters A Comparison of Measurement Approaches" available from http://www.cliftonlaboratories.com/ which enumerates a number of methods. The program implements the 3dB and 45 degree methods in transmission and reflection modes but the 6 parameter HP model is experimental and may give incorrect results. In addition as a variation on the 3dB model, C0 is computed from the series and parallel resonances by the equation

$$C_S = \left(\left(fp / fs \right) - 1 \right) * 2 * C_0$$

To use the program in transmission mode you must connect the crystal in series in between RF out and Detector in, preferably with a jig as shown by Jack and as reproduced here:

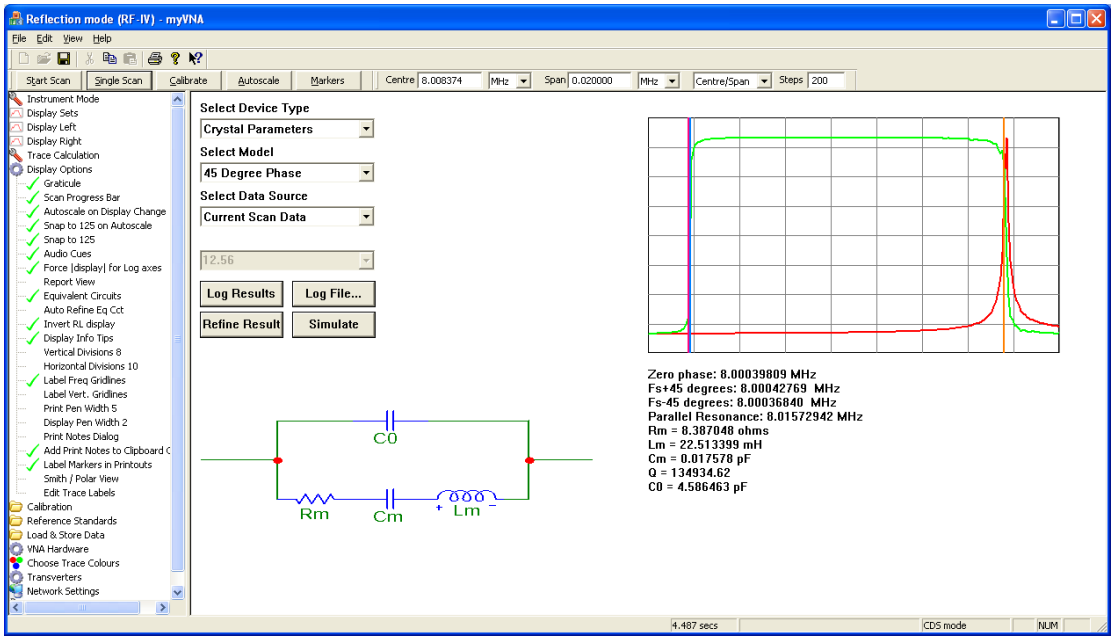


This gives an Rl of 12.5 ohms. Alternatively, you could just use a couple of inline 50 ohm pads. To allow for this the program has a drop down list for selecting Rl, or you may enter your own value if different from the common ones. The program will remember the value you enter. This is only used in transmission mode.

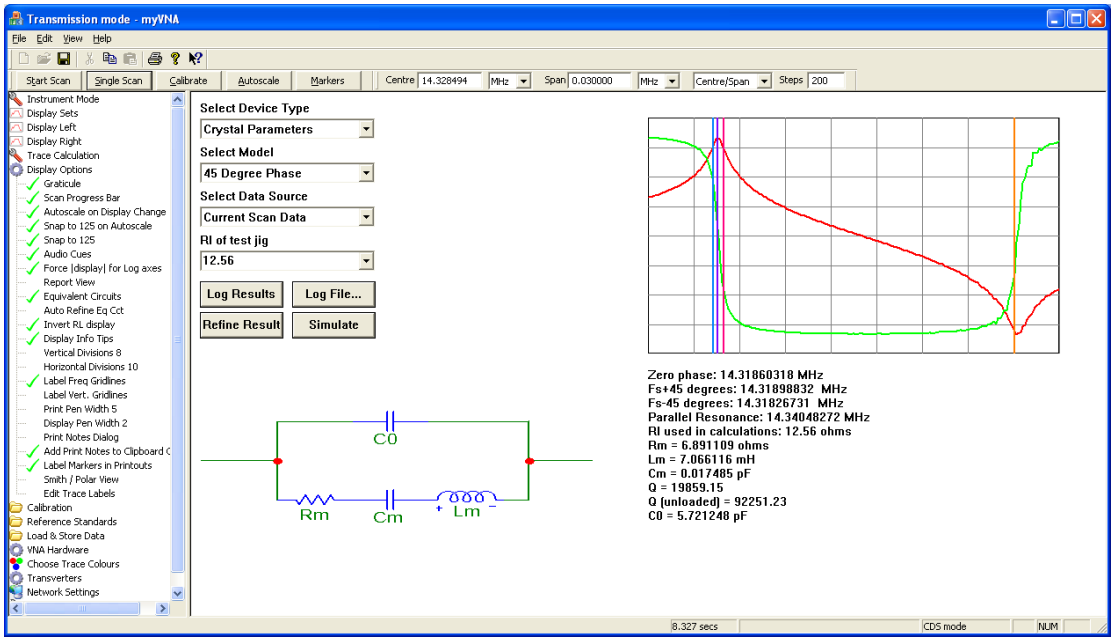
In reflection mode, just connect the crystal to a suitable connector on the bridge between signal and ground.

Before you can use the function, you need to get the display to show the crystal resonances.

Here is what it should look like for the 45 degree parameter model in reflection mode. Note also the displayed results:



And in transmission mode

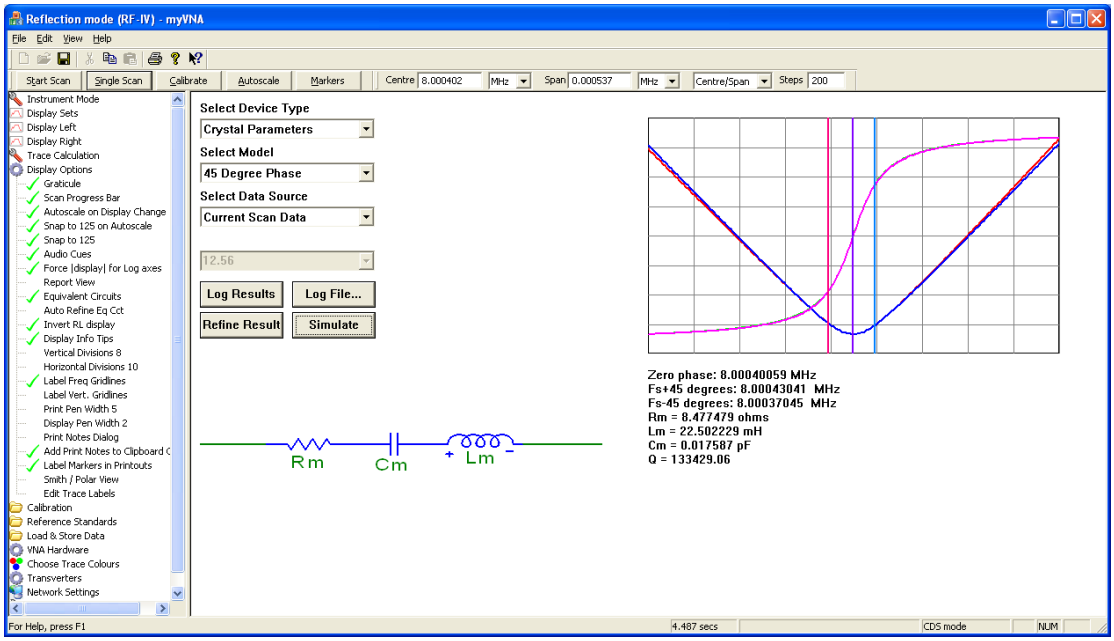


The program will position cursors in the form of vertical lines at the measurement points it has used and display the results of its computation.

If the program detects a parallel resonance, it will display C0 and a 4 parameter schematic; otherwise it will only show 3. Note however that it will not measure parallel resonance alone – the program must have the series resonance within the scan in order to work properly.

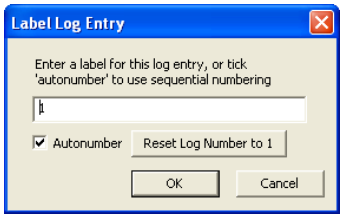
Without using the parallel resonance here is the result. Note that you may zoom in on the series resonance for better resolution. This example is in reflection mode. It also shows the effect of clicking on the “simulate” button. The program has used the network simulation package to simulate the measured crystal; the phase plot is

coincident with the measure results and the impedance can be seen as a red (measured) and blue (simulated) trace.

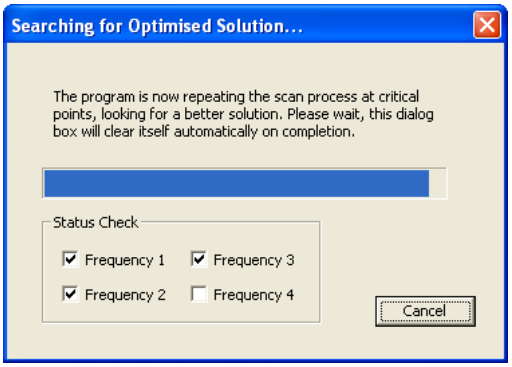


The program operates in both transmission and reflection modes; note that as explained in Jack's memo you need to be aware of the jig's capacitance. Results in reflection mode seem more accurate for reasons not immediately apparent.

To log results, you must first select a log file with the "Log file…" button. Having done that, each time you click on "Log Results" you may choose to log the current readings. A label may be given to each or you may choose to just use sequential numbering, automatically provided.



In most modes, there is the ability to refine a reading. The way it works is this. It may either be switched to automatic mode by clicking on the "Auto Refine Eq Cct" feature on the left view under "Equivalent circuits" or explicitly performed by clicking on the "Refine" button.

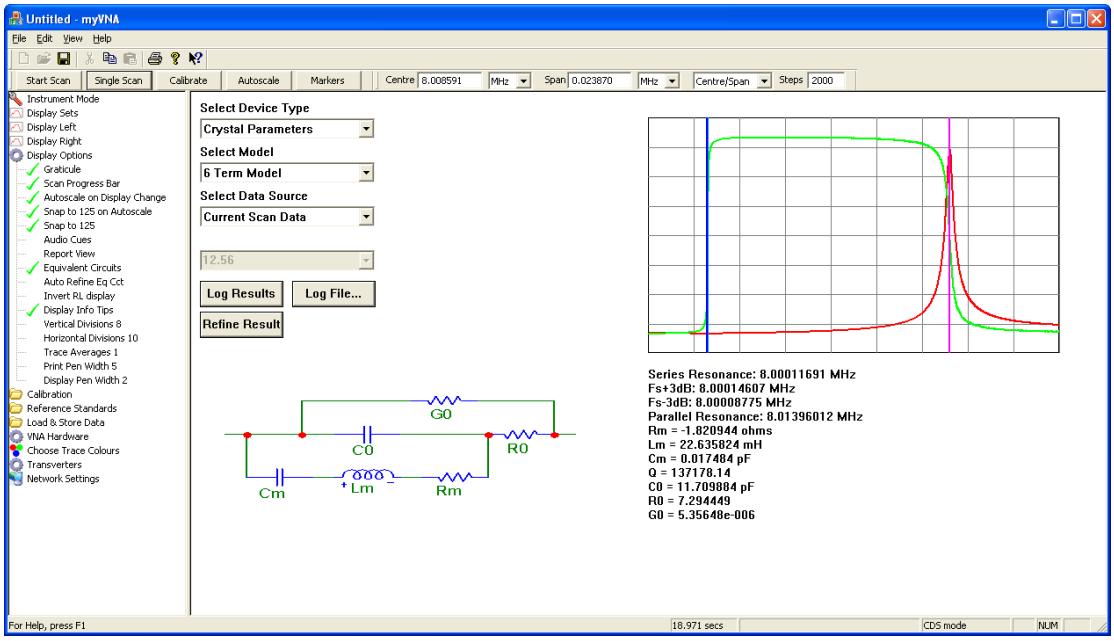

Searching for Optimised Solution...
The program is now repeating the scan process at critical points, looking for a better solution. Please wait, this dialog box will clear itself automatically on completion.
Status Check
Frequency 1
Frequency 3
Frequency 2
Frequency 4
Cancel

When this happens, the program will attempt to locate the best readings for the critical parameters. For each reading that is used in the calculation, it will search for the optimum setting by recursively scanning. The way this works is as follows. For each critical point it will sweep across a range of frequencies from two steps before the point to two steps after it. It will then find the best point in this new scan and again find two points before and two points after and repeat the process, until it finds that the step size matches the DDS step size. It then stops. This means that you can select a fairly coarse scan – provided that the steps are sufficient to give it a starting point – then refine the result. Whilst this is happening a dialog box will show progress. The dialog shows the progress through the parameters by tick boxes (note that for some scans the dialog box may disappear and re-appear) with a progress bar for each scan. The progress bar will be repeated for each recursion performed. Be careful however of a noisy signal.

In manual mode, reselecting the method will revert the results to the original values before the optimisation whereas clicking the scan button will generate a new starting point.

Note that as the program is seeking a solution to a fraction of a Hz, the VNA needs to have reached an operating temperature before the results can be believed.

Finally note that in reflection mode you also have a 6 parameter model available. This is under revision at the moment – don't believe what it says yet. Here is an example of its output:



Read Jack's paper for more information.

When scanning a high Q device like a crystal, some thought needs to be given to the settings. As a starter, I suggest the following but you may find different values work better:

- ADC mode 8 or faster (use 6 for transmission mode)
- ADC sweep start delay 4000 µsec
- ADC step delay 3000 µsec
- Phase step delay 100 µsec
- Initial frequency scan 200 points or more
- Trace averages 5

For a VERY high Q crystal, a more conservative set of parameters is probably called for.

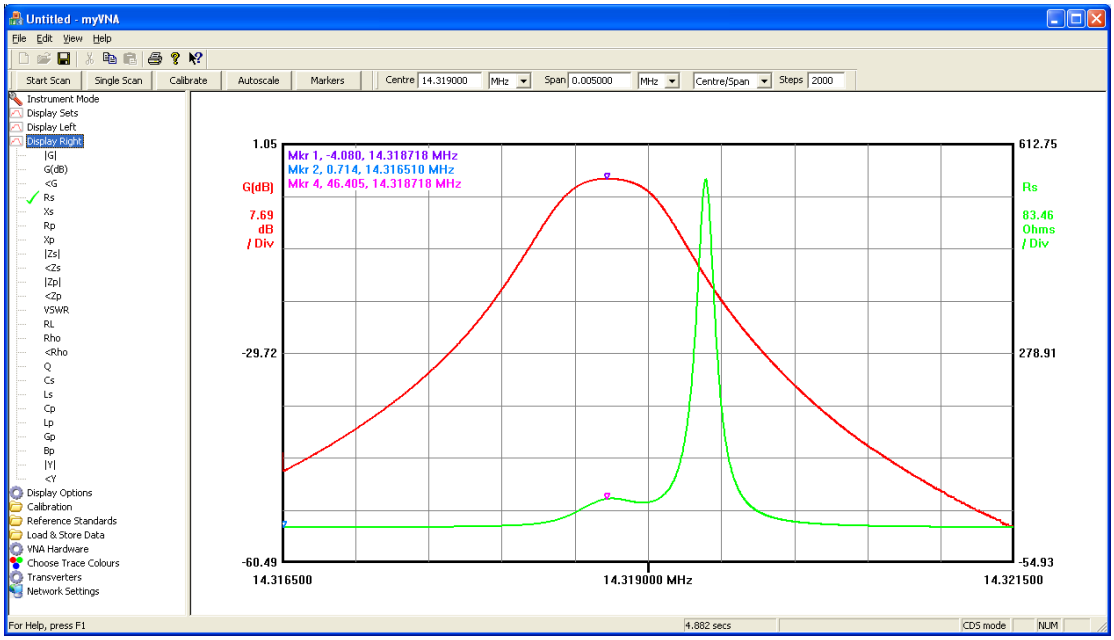
As an experimental addition, there is now a "simulate" button. This takes the current parameters, generates a simulation of the crystal and overlays it on the trace; see the section on Network Simulation for more details. One point to note. If you simulate a crystal that has been measured in reflection mode, then the termination impedance of 50 ohms will give the wrong results, so the program will include a short circuit after the crystal to give the correct reflection (S11) results. Similarly in transmission mode it will set the simulation termination impedance to match that of the transmission test jig (12.56 ohms for example for the jig shown earlier).

11 Dual Detector Scan

Not described yet in this manual, an N2PK VNA may be equipped with one or two ADCs. When only one ADC is present, it may be used for reflection or transmission measurements but not both. When two are present, a scan is usually performed using both ADCs at the same time (A Dual detector scan).

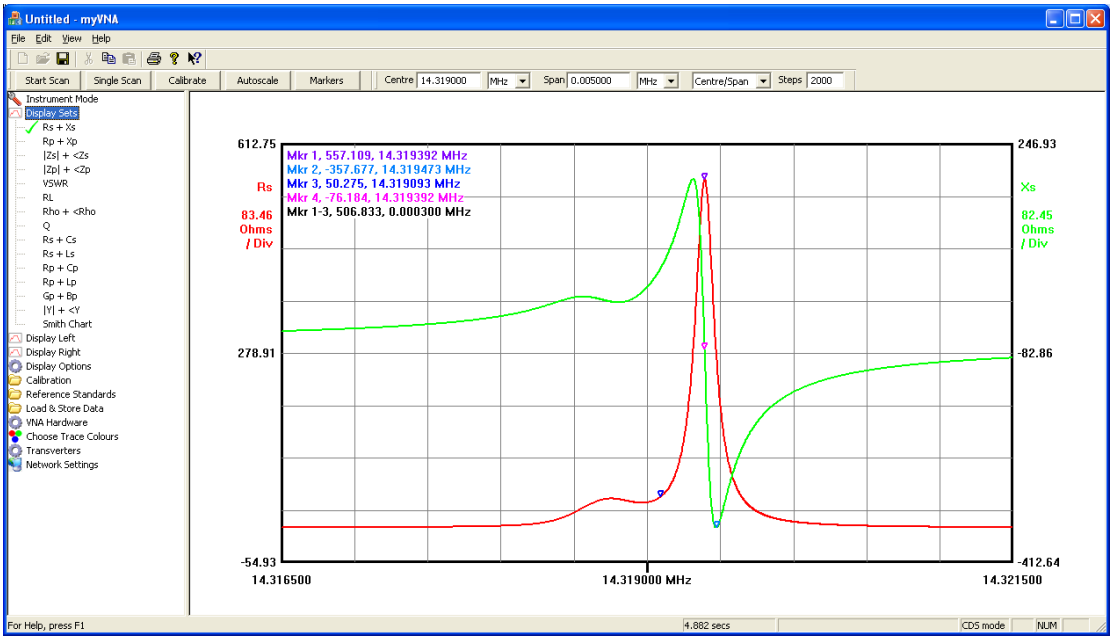
The "Instrument Modes" switches the basic configuration into a number of modes such as reflection, transmission, and dual scan. S Parameters is a variant on Dual Detector and is explained later in this document. If you select Dual Detector, the program will perform a scan using both ADCs with concurrent operation, but to do this it needs to know which ADC is used for transmission and which for reflection, so use the Hardware Options to tell it. Select "Always do Dual Scan" and the program will always do a dual detector scan even if just reflection or transmission is selected.

Having done that, a dual scan trace looks something like this:

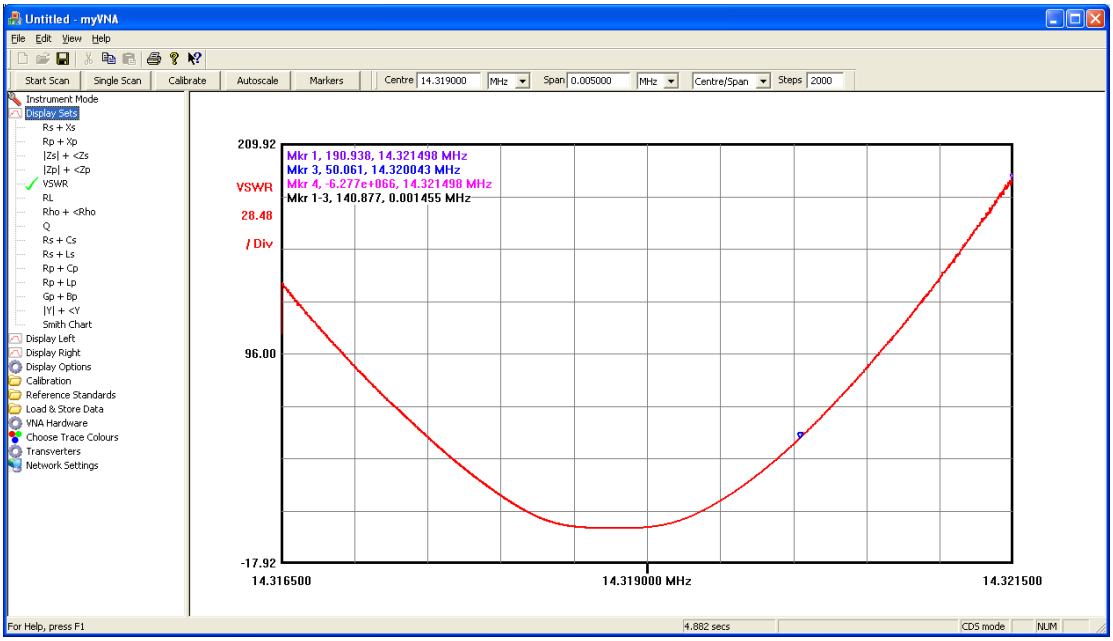


Note that either axis may be set to display any transmission or reflection measurement; in this case this was a scan of a narrow band crystal filter with gain in dB shown on the left (and in red) and the Rs measurement on the right (in green). You may switch either axis to display other readings.

However, remember that the program, once it has captured a set of data can display it in other ways, so if you switch at this point to transmission or reflection modes you may display the data as if you had captured it in a transmission or reflection scan, so for example with the above data set and without rescanning, switching to reflection mode and autoscaling gives this

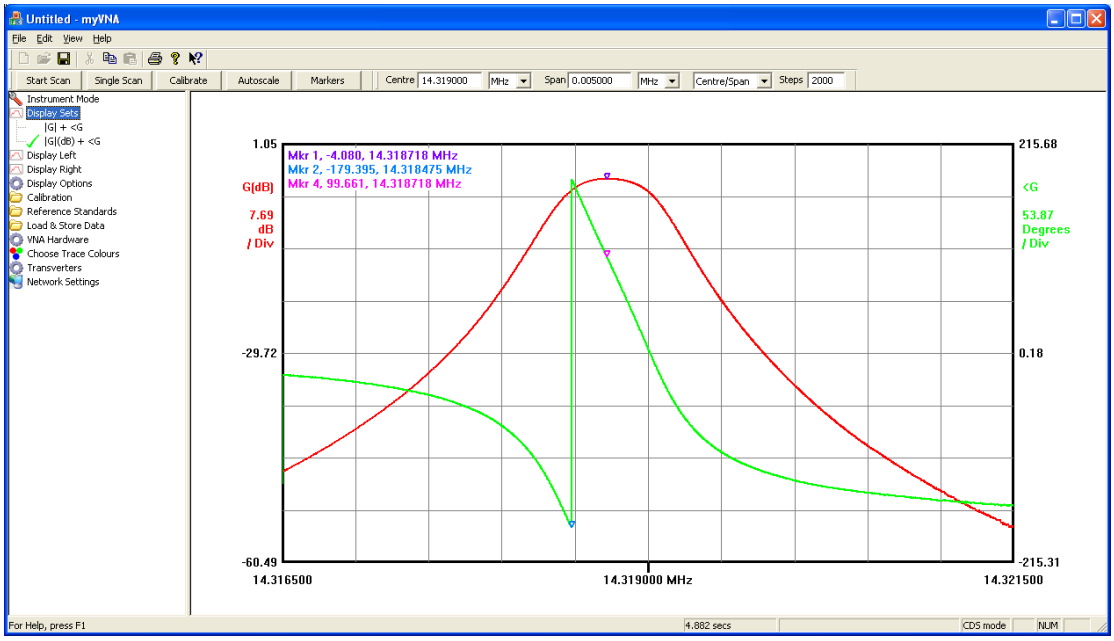


And changing to VSWR gives this



Now it gets fun. At this point, hold down the control key on the keyboard and click on another parameter such as Rs + Xs. The display now shows those parameters too. Click on a few more such as Rho. You can have any combination of parameters you want at the same time with the caveat that the display will get very cluttered.

Switching to transmission mode gives this when gain and phase are selected



This switching function also applies to S parameters. The same multiple display type logic also applies so you may select multiple parameters such as S11, S12, S21 and S22 and display them all at the same time.

12 Logarithmic & Linear Scans

A scan may be performed such that points are linearly or logarithmically distributed between start and stop frequencies. Similarly (but separately) the display may be linear or logarithmic when in rectangular display mode. This chapter discusses logarithmic and linear scans.

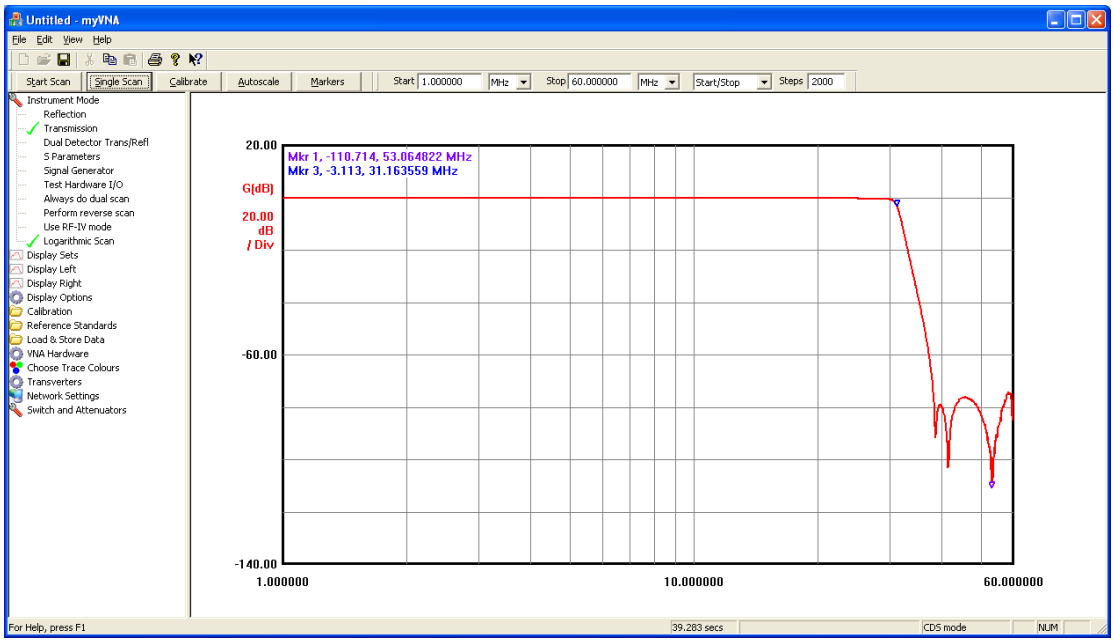
There are two aspects to this; logarithmic or linear calibration data and logarithmic or linear scan data. You may freely mix both in all 4 combinations but remember that at extremes of frequency, especially low frequency, and when using a small number of steps, some errors may be expected when the program generates scan specific calibration data by interpolation. Therefore you may find it wise to use a log calibration scan with a log device scan.

Under “Calibration” is an option for Log Calibration Scans. When ticked, all calibration data collected is done using a logarithmic frequency step scale.

Under “Instrument Mode” is an option for a Logarithmic device scan. When selected, the steps between the scan points will follow a logarithmic scale. Also when selected, it will switch the display to a Logarithmic scale. The display though, need not be the same as the scan; you may select a log scan with a linear or a log display and vice versa.

The mode of the display will change with changes to the log scan, but you may change it at any time by right clicking on the display to pop up the menu, and selecting or clearing “Log Frequency Scale”

The program now checks when it loads saved scan data whether its frequency scale follows a linear or logarithmic sequence (with a certain amount of tolerance). If it finds that the data fits neither, it will not load the data.



All functions, markers, tabular display, file save & load, work equally with log or linear scan data. The only exception is equivalent circuits which wants a linear scan.

When calibrating, the program will also perform a linear step scan – this is purely to keep the code simple. In a log scan, just as when you select a different number of scan points or change the start or stop frequencies, it will generate calibration data for the scan as needed.

Finally, if you select “scan from zero” the program will do its best, but log(0) is not a useful value here, so it will set an arbitrary point at 1Hz.

13 S Parameters

As well as basic transmission and reflection parameters, the program may be configured to work in and display Scattering (S) Parameters. This is the subject of this chapter.

The S parameter model is based on the 12 term error correction model as defined by HP and by following the logic in Paul's code. Refer to Paul's and Tom's excellent papers for details.

In that model, the corrections applied give more accurate results than basic Response Calibration.

There are five different correction models available in myVNA. Response Calibration is the simplest. The 5 and 6 term models assume that forward and reverse connections are identical. This is probably only going to be the case where a single bridge is used and the DUT is manually reversed. In all other cases the reverse direction is going to be significantly different due to bridge and switching differences. Hence 10 and 12 term options are offered. The difference between 5 and 6 term is the same as between 10 and 12 term; in 5 or 10 term these is no isolation measurement of the through detector. It is assumed to be infinite loss.

So, when calibrating choose 5/6 or 10/12 according to your hardware configuration then choose between 5 & 6 and 10 & 12 according to whether you wish to include isolation data.

Note that whatever calibration data is available will then be used optimally in DUT scans.

The best results for S parameters will be obtained when forward and reverse scan data is available. This is not the same as whether you performed 5/6 or 10/12 term calibration. In a reverse scan, if 10/12 term data is not available, the 5/6 term data is used in both directions.

How do you make the program do a reverse scan? There are two ways. Manually, there is a tick option under "Instrument Mode" called "Perform reverse Scan". When this is ticked, if appropriate, the program will assume that the data it is reading is reverse direction data and process it accordingly. However, if an S parameter test set is being used, the program can automatically control the relays that switch the test set. To enable this, tick one or other of the "Enable Switch X in Reverse scan" options under "Switches and Attenuators". When one of these is selected, the program will automatically perform a reverse scan after a forward scan.

A word of caution, it is very easy to get confused about data in forward and reverse scans. If you perform a reverse scan only and display a forward scan data item, or conversely do the opposite, then when scanning you will see a 'trace in progress' indication but no trace data will be shown at the end.

Note that for the parallel interface there is just one switch line. For the USB there are two, the program will automatically reverse scan if either is selected. Note that if you tick the second (USB only) switch line then select a parallel interface, the second switch line will be hidden – which can be confusing as it will still cause automatic reverse scans.

In summary then, when you perform a dual detector 12 term calibration scan, the program will need to capture the following:

1. During the Open scan it measures transmission isolation and reflection coefficient for "open"
2. During the Short scan it measures reflection short data
3. During the Load scan it measures reflection load data
4. During the Through scan it measures through transmission data and also measures the reflection coefficient of the detector.

These 6 factors are applied to forward and reverse data giving 12 terms. The program may also be set to use 10 term (in which case the Open scan does not measure Transmission Isolation) or 5 and 6 term ERC. The 5 and 6 term models are like the 10 and 12 term models but the reverse scan calibration data is assumed to be the same as the forward scan data. In 5 or 6 term it is therefore a big assumption that the program is making that the cabling or relays used to perform reverse measurements do not introduce an error, but if you are only performing forward measurements (or manually reversing the DUT) then a single bridge configuration may be used. See Annex B for more information on test sets.

The type of calibration scan is explicitly set in the "Calibration" treeview control. Set it to Response Calibration or one of 4 other Calibration modes (5,6,10 or 12).

The sequence in which it will ask for these 5,6,10 or 12 data items will vary by the mode selected and by whether the program has control over the forward / reverse settings. It is preferable to use an S parameter test set and automate the switching. The program will know you have an automated test set if it sees that the switch line (parallel) or either switch line (USB) are set to change on S parameter switch control (in the Switches and Attenuators set of controls). The sequence for a manual scan is simple – do the forward then (if 10 or 12 term) the reverse data set. For automated switching however, the sequence tries to optimise the handling of the open, short and load standards to make it as easy as possible. Take care however to read the instructions on screen as it is not immediately intuitive.

During any scan including the calibration scans it is possible to average a number of traces (up to 10,000 per point if you want to go mad) but during the Open / Isolation scan, it is possible to configure the program to average a different number of points. This is useful, for example, in increasing the accuracy of the low signal level isolation scans without incurring the overhead of excess averaging at other settings. This is controlled by the "Use separate isolation average" option and "Isolation Average" count in the "calibration" node of the treeview on the left.

Calibration data can therefore be captured in a number of ways:

1. Reflection scan. Open/Short/Load reflection data only are captured
2. Transmission scan. Through and optionally isolation are captured. Isolation data is set to zero if skipped. Through reflection data is set to zero always.
3. Dual Mode or S parameter. As defined above, this captures the reflection data, transmission data and detector reflection coefficient for 5/6/10/12 term correction.

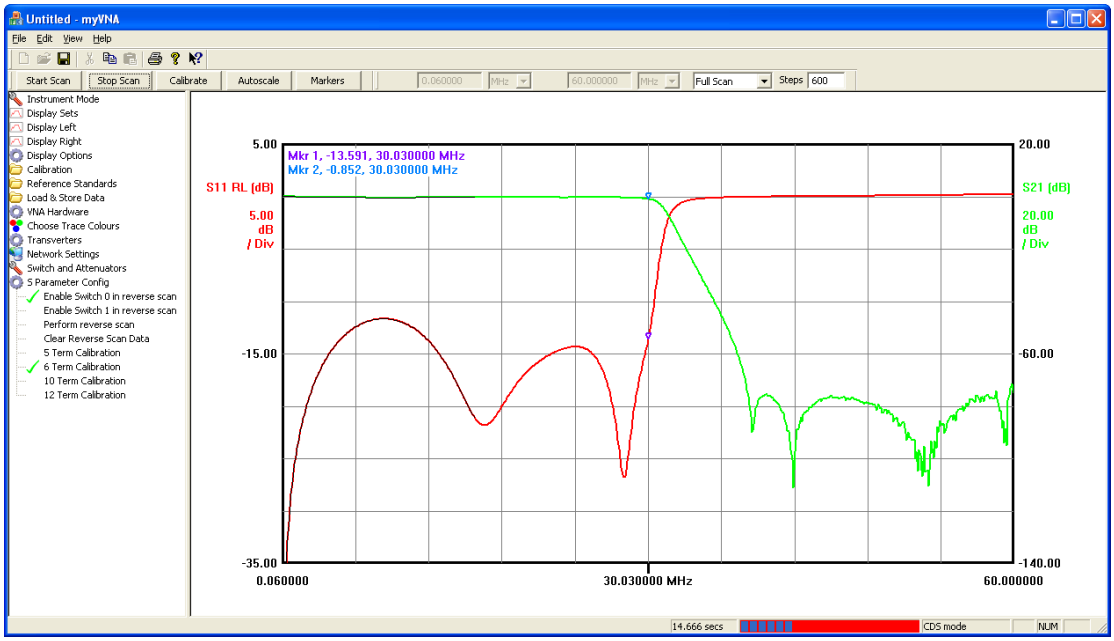
Therefore if you calibrate separately for reflection and transmission, detector reflection coefficients will be missing from the calibration data and therefore a source of error in the S parameters. The program will not stop you using such data (you might deliberately intend such a measurement).

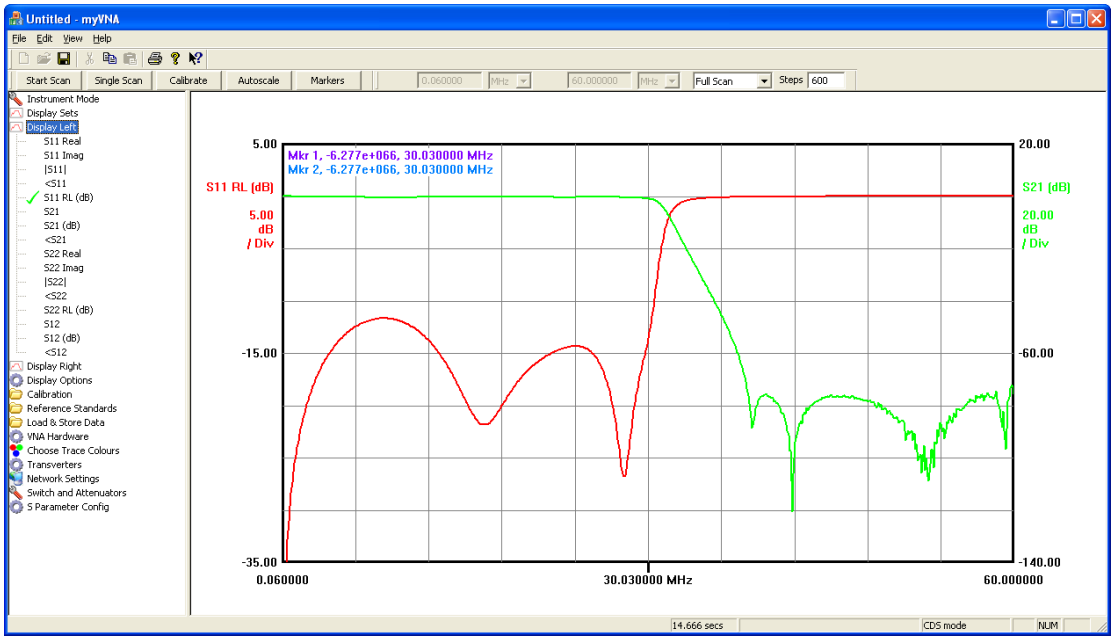
Note that under “Hardware” is an option to set an initial scan delay that may be used, to take account of relay switch times. Also remember that only one switch line is available with the parallel interface, two with USB.

For a full 2 port S parameter scan, a set of forward and of reverse data are needed. If you do not perform a reverse scan but only give it forward data it will still function but the return parameters in the calculations will be set to zero. The results are not as accurate as full correction with 10/12 term.

When a set of S parameter data has been obtained, you may switch to the other instrument modes – basic dual detector, reflection or transmission.

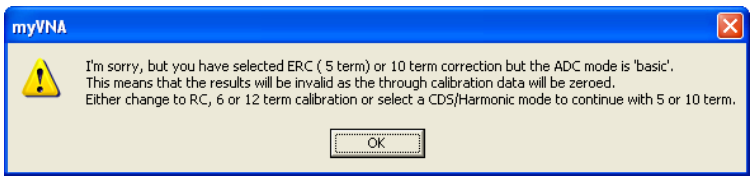
When the program is scanning, it will turn the colour of the progress bar red whenever it is doing a reverse scan as shown below





And above here are the display options for S parameters currently configured

There is a limitation to note. In CDS mode, the software eliminated detector offsets so with no signal the detector should read zero. This means that we can safely use 5 or 10 term correction, where we assume that the through isolation is zero. In basic mode though, the detector offset is not removed so it is not safe to use 5 or 10 term calibration. You would get the wrong results, so it will not permit such a combination and will give the following error message.


myVNA
I'm sorry, but you have selected ERC (5 term) or 10 term correction but the ADC mode is 'basic'.
This means that the results will be invalid as the through calibration data will be zeroed.
Either change to RC, 6 or 12 term calibration or select a CDS/Harmonic mode to continue with 5 or 10 term.
OK

Under ‘Hardware’ is an option “Swap detectors in reverse scan”“. This should be ticked if the function of the detectors reverses for reverse scans. Only tick it if this is the case. Get it wrong and the results will be weird. A “T check” is a good idea when commissioning a system. Perform a calibration (5,6,10 or 12 term, it does not matter) then connect a 50 ohm load to a T piece in series with a through connection. The result should be an S11 and S21 measurement that are flat to within about 0.1dB across the frequency range.

So far, there is one limitation not addressed. The reflection data display (things like Rs+jXs) only worked for Port 1. There is an option under “Instrument Mode” that allows the port that is used for the reflection data display to be changed from port 1 to port 2.

14 Harmonic modes and CDS

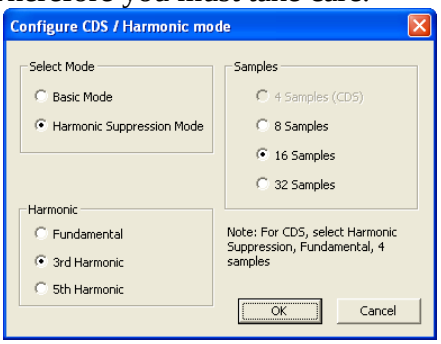
Paul Kiciak and Wayne Torrey have developed a method for managing harmonics in the N2PK. This may be used in two ways

1. To suppress harmonics that cause strange effects at high frequencies
2. To extract a harmonic (2^{nd}, 3^{rd}, 4^{th} or 5^{th}) to extend the frequency range.

This version includes an experimental implementation of the algorithm, courtesy of Paul. This chapter presents details on its operation.

Use it with care, and ONLY if you understand how to use it. For this refer to Paul's notes on the Yahoo group.

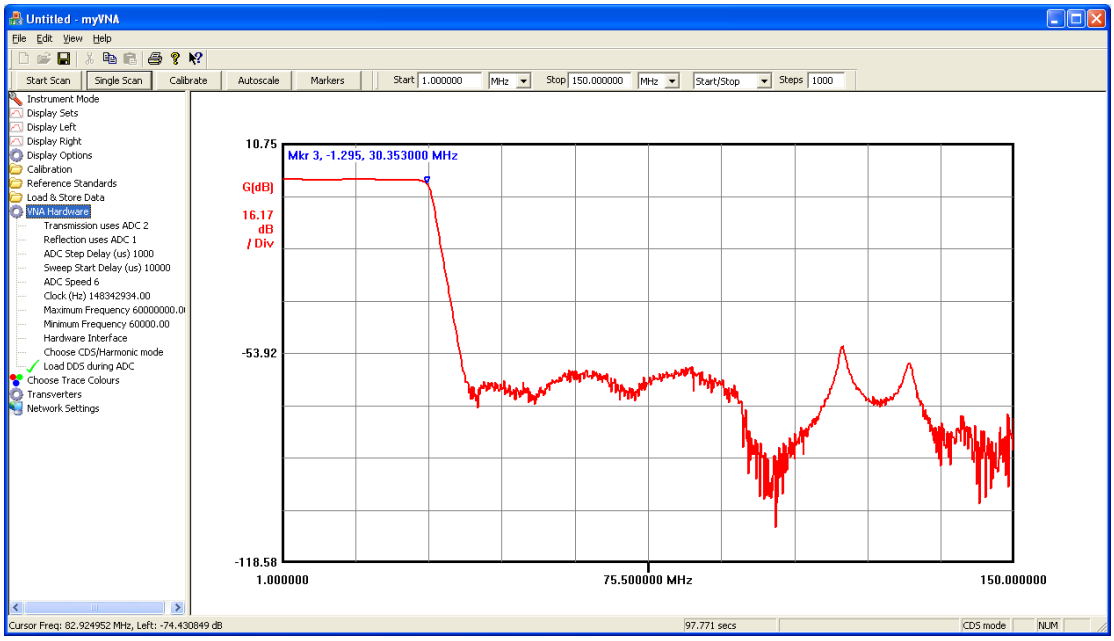
The program ought to limit calibration data so, for example, it should not allow you to use harmonic calibration data in another mode. This it currently will not do as the modes are experimental. Therefore you must take care.



In this example, the program will extract the 3^{rd} harmonic using 16 samples per step (so it gets slower).

When you select 3^{rd} or 5^{th}, a multiplier is applied to the frequency settings. Note that you should NOT change the minimum and maximum frequencies – the program will automatically apply a multiplier to the frequency scales used.

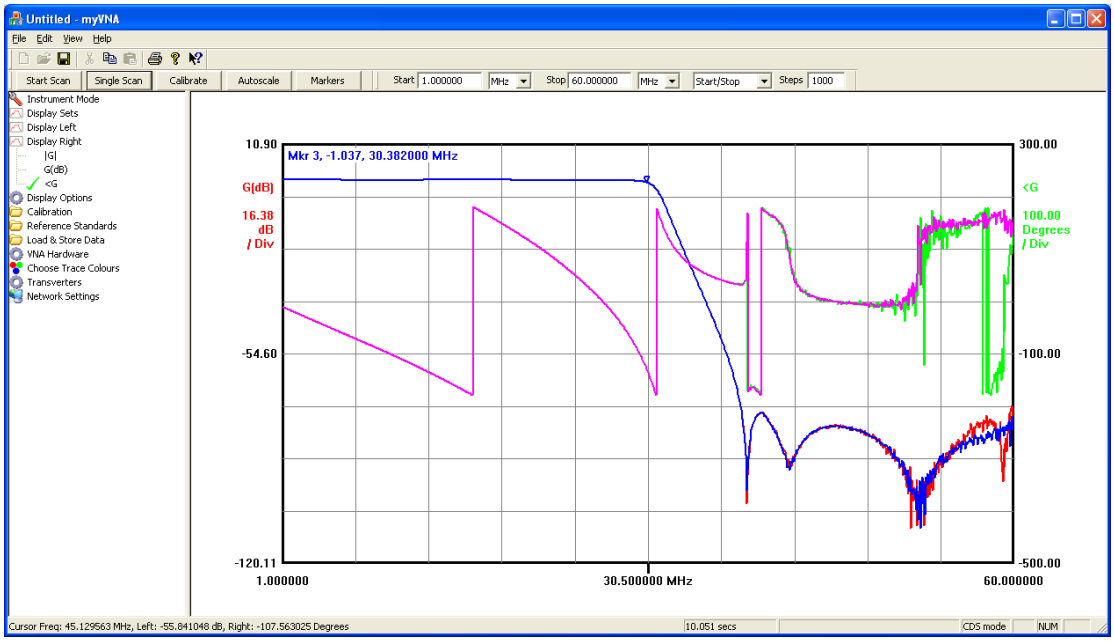
In 3^{rd} harmonic mode for example the scan range of the VNA changes to 180 KHz to 180 MHz. Here is a scan of a 30 MHz low pass filter from 1 MHz to 150 MHz:



To use it, you MUST use a harmonically rich RF source as described by Paul such as an AC series gate biased to its mid-point.

Look also under ‘Calibration’ and you will see an option “Relax Harmonic Mode Checking”. Normally the program will not allow you to use calibration data with one harmonic mode to scan in another mode. Under certain circumstances this may be useful; tick the menu item to permit it.

Harmonic suppression is also useful at baseband. Here is the same filter from 0 to 60 with normal CDS mode. The red & blue traces show the gain differences with harmonic effects at the upper end of the range, and the purple / green show the phase differences. Green and red traces are normal CDS mode.



15 Group Delay

Within the transmission modes there is a Group Delay option. Selecting this will cause the program to display group delay by calculating

$$delay = -\Delta(phase) \Big/ \Delta(freq)$$

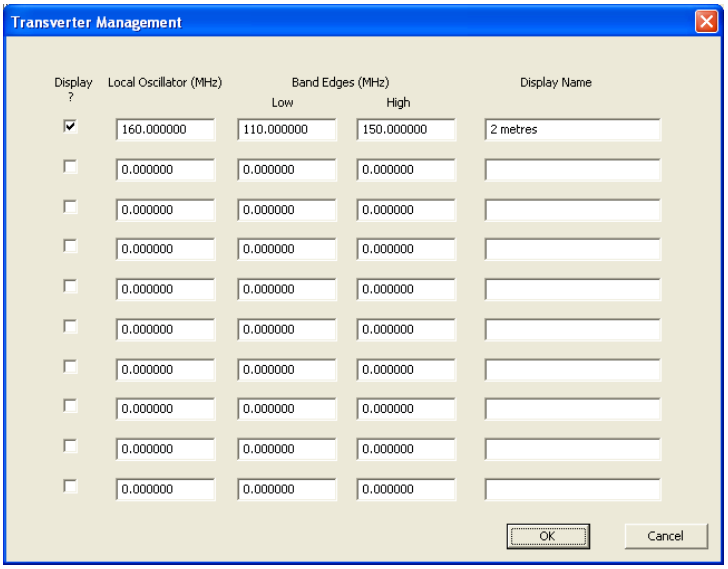
Where the readings are from adjacent scan points. This is not ideal, it would be better to use a frequency aperture but that is not implemented. The implication is that you must ensure that there are sufficient scan points to ensure no big phase changes between points, or accept that the result will be in error at that point. Because it uses scan data, there is no measurement for the first point in a scan, rather the program will display the same reading for the first two points.

16 Transverters

External hardware transverters may be used to extend the VNA operating frequency range. This chapter explains how to configure the program to use them.

When you open up the transverters node, the only items shown will be “Configure Transverters” and “none”.

“Configure Transverters” will pop up a dialog as shown below.



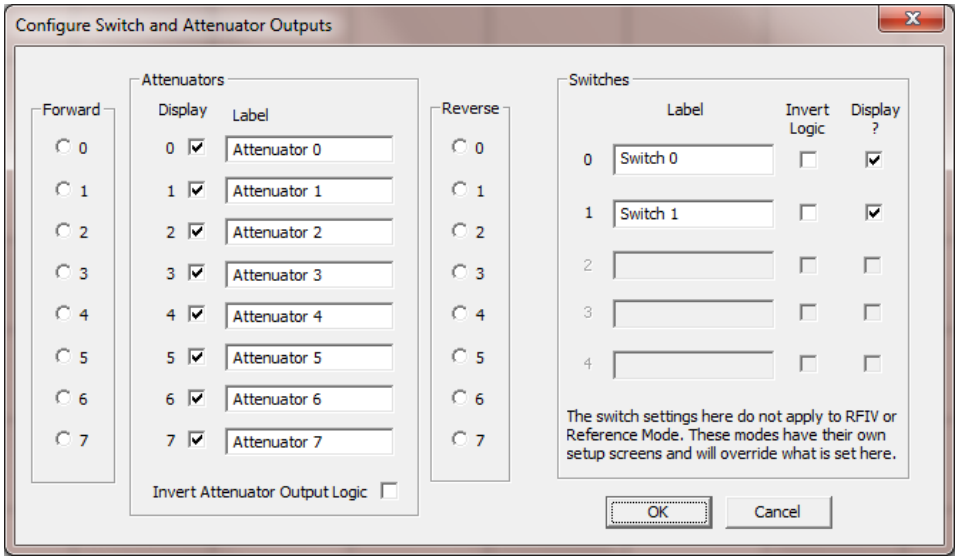
This may be used to configure up to 10 transverters. Every one for which a tick is shown in the “display” column will appear under “none” in the transverter list and may then be selected for use (a green tick will appear next to the currently active transverter, or against “none”).

For each displayed transverter, the program needs to know the local oscillator frequency and the actual band edges for the transverter. It can be either side of the LO, but must be wholly on one side or the other. myVNA will figure out the correct scan direction based on whether the band given is below or above the transverter LO. Finally a display name of up to 20 characters must be entered – this is what will be shown in the list of active transverters. The transverter data will get stored in the registry on exit and automatically reloaded on startup.

17 Attenuator and Switch control

Switch and attenuator control functions are present to support S parameter measurements and to permit external scan synchronisation should it be needed. This chapter addresses how they are configured and used.

On the left is a new treeview item labelled “Switch and Attenuators”. Open it up and a number of items are displayed, the top one of which is “Configure settings”. This opens up a dialog box that allows the functionality of the control lines to be set.



On the left is the attenuator control. You may select which of the 8 possible settings are displayed in the treeview and the names associated with them. You may also ask the code to invert the logic of the output lines to suit your hardware.

On the right will appear one or more switch controls (The second is only available with the USB interface). Again for each the display name may be set, it can be excluded from the treeview and its logic may be inverted.

Having configured the settings, clicking on an item in the treeview will select that option and place a tick next to it.

On the left and right of the main block of attenuator controls are two columns with radio buttons labelled “Forward” and “Reverse”. When the program is configured to automatically switch the attenuator before each scan, the values will be set to the chosen settings depending on whether the scan is forward or reverse. When enabled (by ticking the “Set Attenuator Automatically” treeview item, the manual settings for the attenuator will be overridden by the appropriate automatic settings at the start of each new scan. This functionality is designed to be used in S parameter amplifier testing where different attenuator values may be needed in each direction for best performance.

17.1 Toggling switches during a scan

Under “Switches and Attenuators” are two options labelled “Invert Switch 0 during scan” and “Invert Switch 1 during scan” which when selected cause the corresponding switch line to be inverted at the start of a scan and reinstated at the end. This may be used to synchronize external hardware to a scan. Because it uses the same switch lines as other functions such as the RFIV control, it is only active when such functions are not active.

18 Open/Short/Load Calibration Standards

Reference standards are critical to the operation of a VNA. This chapter explains how the program uses them and how their characteristics are made known to the program.

On the treeview is an item labelled “Reference Standards”. If you open this up you will find a set of entries for the Open, Short and Load standards plus the option to save and load standard data.

The program maintains a notion of the errors inherent in the calibration standards and corrects for them at the end of a calibration scan and when reloading calibration data from disk. The parameters may be changed as you see fit within certain limits, and it will remember the values when the program exits. You do not need to explicitly save them when they are changed. To change a parameter, select it from the treeview, and a dialog box will appear inviting you to change its value. Limits apply to each and these will be indicated in the dialog box. The program assumes that an open suffers from fringing capacitance, a short from a series resistance and impedance and a load a resistance and inductance in series with a capacitance in parallel with them.

18.1 Loading & Saving Calibration Standards Data

You may want to be able to switch between different calibration standards however, and for this reason the complete data set may be changed by saving to file or loading from a file. This file is a text file and may be edited as you wish as long as the format is maintained.

It is important that you understand when the correction factors are applied. When you perform a scan, the correction factors for the OSL standards will be applied automatically at the end of the calibration scan, so they must be set correctly before calibrating.

18.2 Saving Calibration Standards in Calibration data files

Under “Calibration” are two extra options – “Save Mode with Cal Data” and “Load Mode with Cal Data”. This permits the instrument mode, the dual scan option, the current attenuator setting, S parameter switch settings and reverse scan flag to be saved along with the calibration data. It may then be automatically loaded again whenever the calibration data is loaded. It is recommended that this option be used. If not, care must be exercised in loading the correct standards data before loading a calibration data file generated with that standards data.

18.3 Saving System Reference Impedance in Calibration & Standards Files

There is the option to save & load the system reference (Z0) from standards data and calibration data files. This only applies to files where Z0 is actually stored – in other words files saved by software version 86 or later. In these cases Z0 is automatically included in the files and there is an option (under the “Hardware” section called “Reload Z0 with calibration data”) to set the system reference when the files are loaded. Untick it to stop this happening.

18.4 LLC (OSLC) Calibration

Low Loss Capacitor calibration is also provided. This results in a 4th step to the normal OSL calibration – call it OSLC. Don't use it unless you understand it. The 4th step uses a high Q capacitor whose characteristics must be known to the program. This is set by entering the value of the capacitor, its Q at a given frequency, that frequency and the model to be applied. There are 3 models for now, fixed Q, an equation based on a high quality vacuum capacitor and one based on a porcelain ATC100B capacitor.

18.5 Calibration Notes

It is possible to put calibration notes into a calibration data file. If enabled (the function is disabled with a tickbox under View / Options) then every time a calibration data file is saved to disk, a dialog box will pop up first with the current calibration notes. You may clear the notes, cancel the inclusion of notes or include the notes in the calibration data file. When calibration data is loaded, the notes are displayed at the bottom of the calibration data summary dialog.

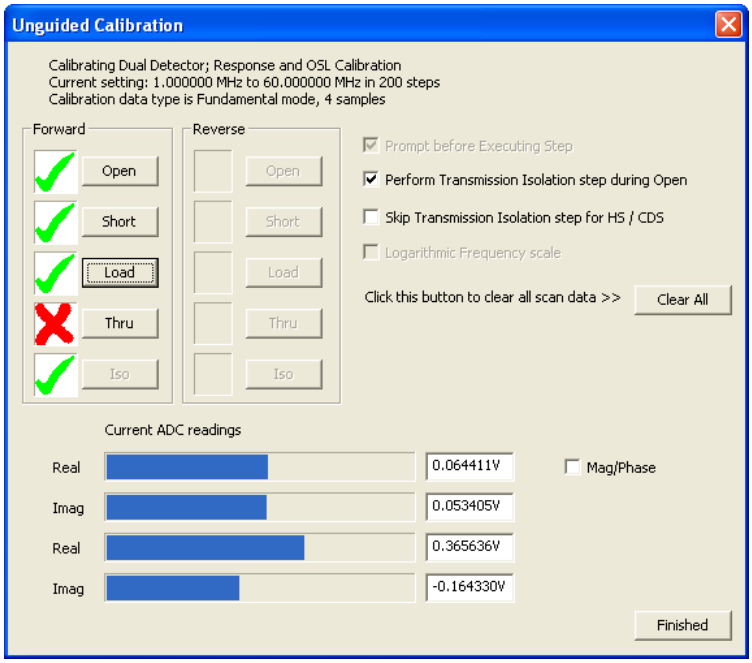
18.6 Loading / saving settings in calibration data

Under the 'Calibration' options is a selectable item "Load mode with Cal data" and under View / Option is a selectable option "Save basic mode data in calibration data file'. This controls the saving of basic scan parameters in a calibration file and whether, when the file is loaded, the settings are applied.

19 Unguided Calibration

There are two modes of operation for calibration – guided and unguided. This chapter explains the unguided mode. The guided mode was shown in the Tutorial chapter.

It is not always the case that you want to follow the step by step guided mode through calibration. When you get used to the program you may want simply to 'do your own thing' in regards of the sequence of steps and do not need the prompts. For this there is an 'unguided mode'. This is presented as an option under 'calibration'. When selected it changes the mode of the calibration button. Clicking 'Calibrate' will give the following dialog



Note the three lines at the top of the dialog showing the type of calibration and the current settings. Initially it shows the current calibration data. If you perform a calibration step it may change as the step will be performed to the current calibration settings and not to the current calibration data settings.

The type of calibration (6 term ERC in this case) is taken from a combination of the calibration mode selected from the Calibration settings (Response / 5 / 6 / 10 / 12 term) and, if Response Calibration is selected the Instrument Mode. Response Calibration changes depending on whether Reflection, Transmission or Dual Detector mode is chosen and whether the 'Always do dual scan' option is selected.

The Current Settings are taken from the current calibration data "open" settings if there is any valid data present. For each parameter, a green tick or red cross is displayed if the calibration data for that parameter appears valid or not.

Note that the current VNA frequency and scan point settings may be different to the displayed value if the calibration data does not match. As soon as you calibrate any parameter, it will use the current VNA settings for that parameter and adopt the new

settings. Because of this, a set of green ticks can change to a set of red ones when a new scan is taken.

There is one button and one cross/tick for each parameter. All that are currently valid will be enabled. The Isolation step though merits discussion. Whether it is enabled or not depends not only on the calibration mode but also on the two tick boxes on the right. In Response Calibration when doing a Transmission scan, you may choose not to do an Isolation scan if a CDS or Harmonic mode is being used. When available, the tick box will be enabled and ticking it will grey out the Iso button and the Isolation data will be zeroed when the Thru calibration is performed.

The other tick box that may be shown is labelled ”Perform Transmission Isolation During Open”. Again, if ticked, the Iso button will be greyed out. Now, the Isolation data (i.e. transmission Open data) will be collected when doing the Open scan.

Always shown is a tick box that allows override of the log/linear calibration scan setting.

The tickbox labelled “mag/phase” toggles the display of the current ADC readings between real & imaginary and magnitude & phase.

20 Reference Planes

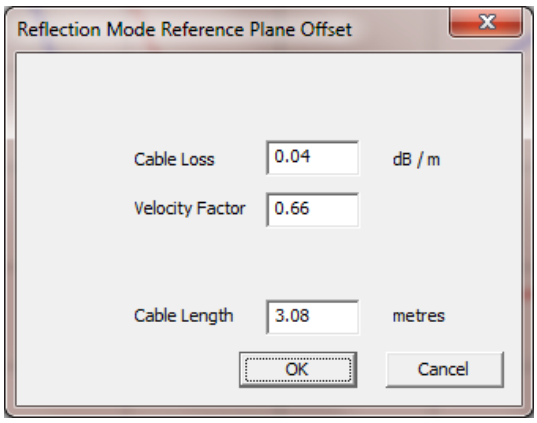
An offset to the reference plane may be entered using the control “Ref Plane Offset (mm)” located under the Reference Standards control.

This offset is designed to be used to move the reference plane by a small amount, for example to move if from the back of an SMA to the mating surface of the connector. The implementation is of a lossless perfect line and the value required is the number of millimetres; a positive value towards the VNA and a negative value away from it.

Specifically the complex reflection coefficients measured for each of the OSL values is multiplied by

$$(1 - j * \tan(2 * \pi * L / \lambda)) / (1 + j * \tan(2 * \pi * L / \lambda))$$

It is also possible to apply a transmission line correction to measurements in reflection mode. The controls for this are located under “Trace Calculation” and comprise a Boolean flag to switch it on or off and a dialog to configure it as shown below:



The cable loss does not take account of frequency so this will be approximate. When active, the reflection mode measurements (not transmission or S parameter) will be modified. Specifically the complex reflection coefficient measured by the VNA is modified by multiplication with the value below

$$\frac{(\sinh(\gamma L) + \cosh(\gamma L))}{(\sinh(\gamma L) - \cosh(\gamma L))}$$

Where L is cable length in metres and $\gamma = \alpha + j\beta$

and $\alpha = CableLoss(dB/m) / (20 * \log 10(e))$

and $\beta = (2 * \pi) / (VelocityFactor * \lambda)$

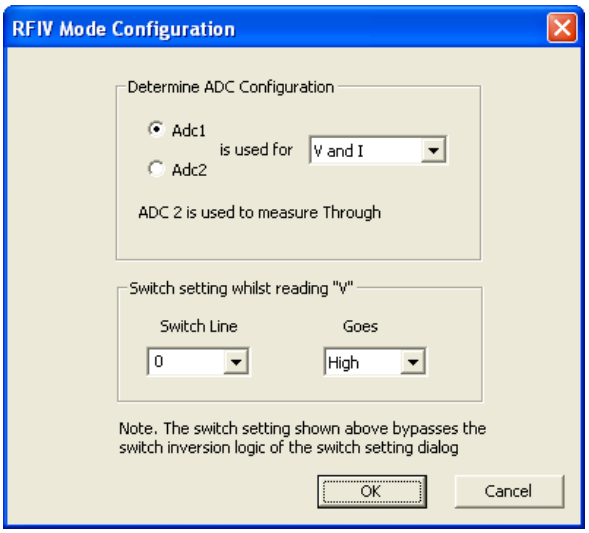
The corrections apply, when enabled, to stored data as well as to the results of the current scan. All reflection measurements not just the reflection coefficient are modified (Zs for example).

21 RF IV Method

The RF IV method uses a specific hardware interface to perform accurate impedance measurements. Here, measurements of voltage and current are made separately and their ratio used as a measure of impedance. RFIV is not available on the parallel interface.

The implication of this is that measurements of V and I are both needed. One way to do this is with a dual detector configuration, where one detector is used for each. Another (and more accurate) method is to use a single detector, but this means a fast method of switching is needed and the process will take twice as long. Both options are supported by the software.

Under “Hardware” is an option to configure the RF IV mode of operation. Selecting it will open the following dialog.



The setup screen permits choice of which ADC(s) are used for V and I. This also determines which ADC is used for the through measurement. The software functions almost identically to Reference Mode, the difference being that the through signal is not divided by R.

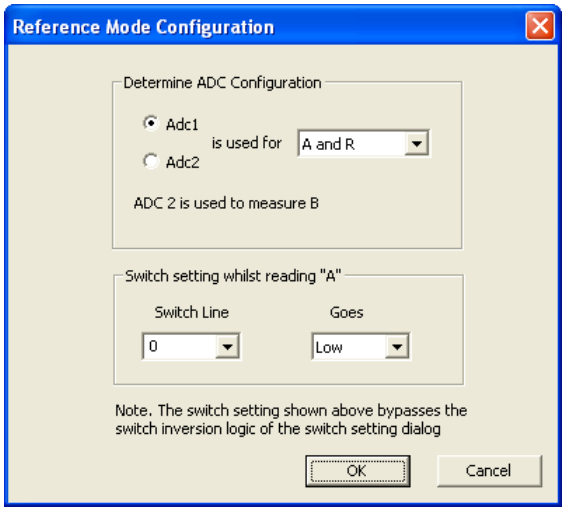
To enable the RFIV mode, select “Use RF-IV mode” under “Instrument Mode”. When this is selected, the scan mode changes to Reflection and the calibration mode to “Response Calibration”. You will need to perform a specific calibration process with the RF-IV hardware, and this may be saved as a calibration data file. The program will store along with the calibration data a flag indicating whether the calibration data is RFIV or not and will not allow you to mix them. When the program starts and it loads its last set of calibration data, if it finds that that data is RFIV mode data it will automatically switch to RF-IV mode.

RF-IV mode, like Reference Mode supports transmission mode, but it will not give the right results. It has been left available for experimental purposes, but you should not use it for other than reflection mode measurements.

22 Reference Mode

In a similar manner to RF-IV mode, reference mode uses an external accessory for higher precision measurements. This chapter describes it.

Reference mode is supported by means of a configuration screen and an option under “instrument mode”. Currently Reference Mode is not available on the parallel interface. The configuration mode setup dialog looks like this

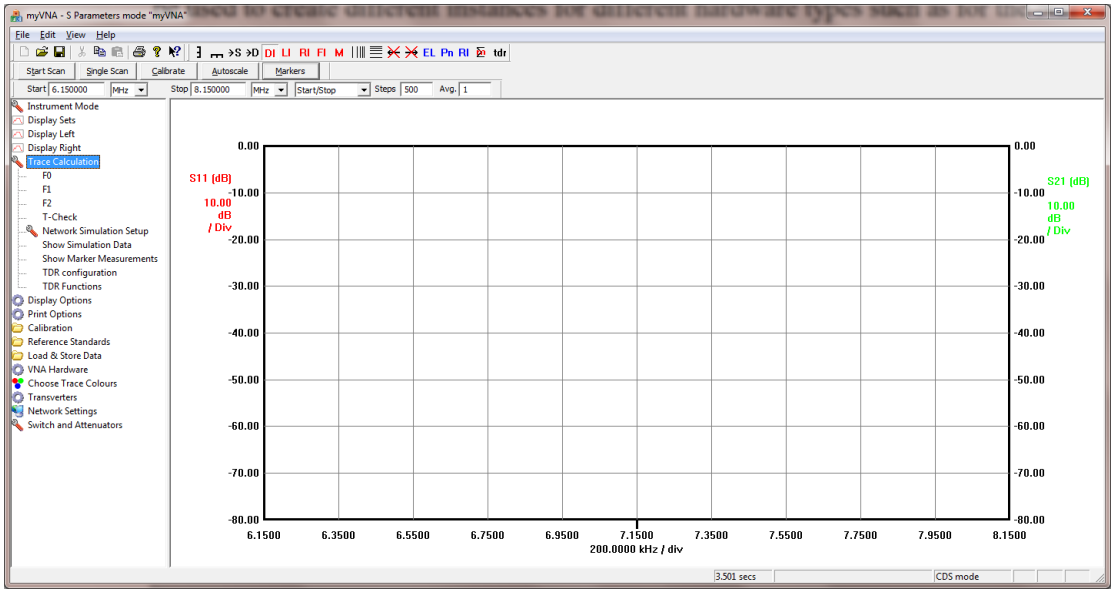


The hardware settings must match that of the hardware. If you use one of Paul’s modifications to Ivan’s RFIV head, then one ADC will be used for A and R, the other for B, where A is the reflection signal, B is the transmission and R the reference. So the obvious configuration is to use ADC1 for A and R as shown above. The switch line to use depends again on hardware; with USB you have two lines to choose from. Whichever you use the sense must match the hardware. Again, with Paul’s modification to Ivan’s board, connecting one of the USB switch lines direct to the switch control line of the Reference mode head means that to measure the ‘A’ signal the switch line must be low.

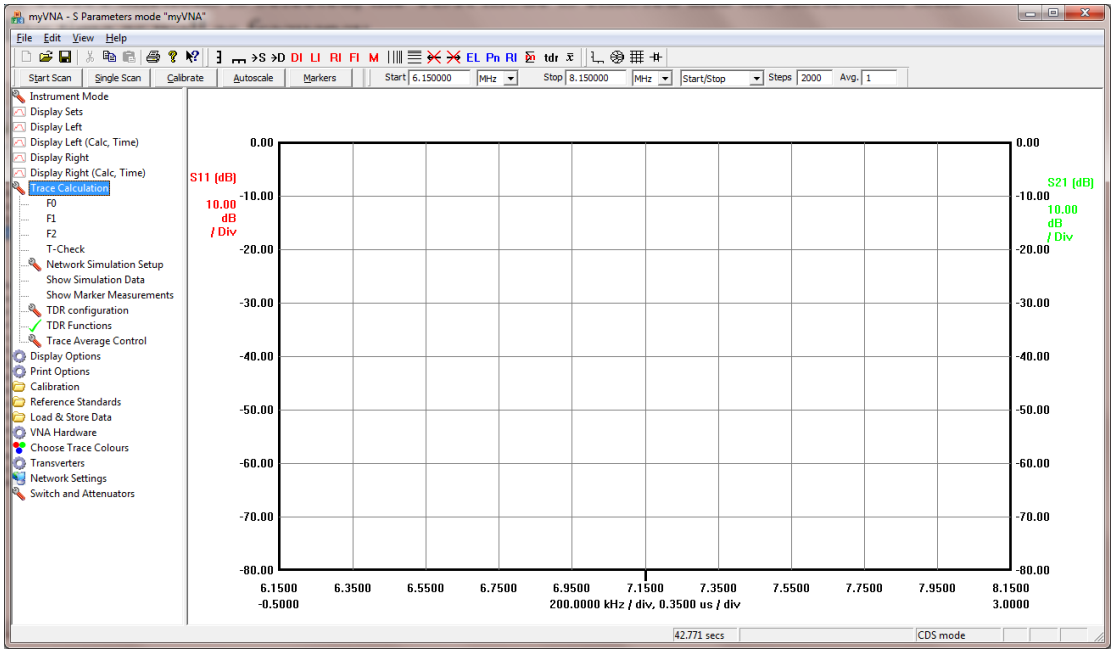
23 TDR mode

Data captured in the frequency domain may be converted to time domain in software to give a Time Domain trace. This chapter describes it.

Look under “Trace calculation”. There are two TDR options. There is also a new toolbar button for access to the configuration option.

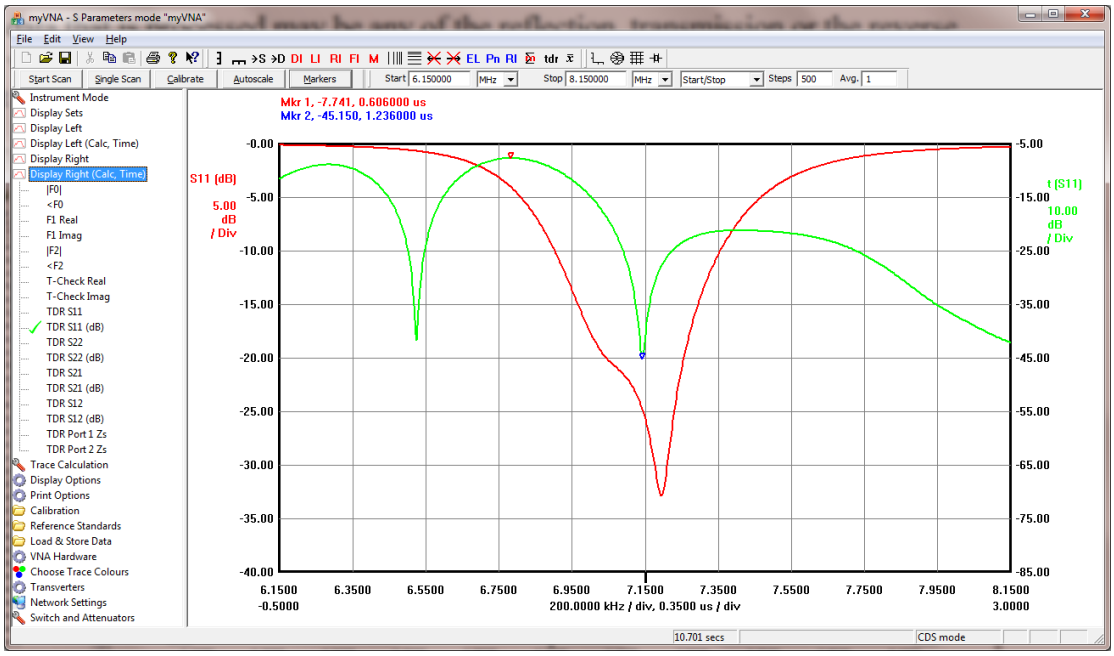


When TDR Functions is selected, the TDR mode is enabled and the horizontal axis displays time as well as frequency.



What the program does is to take a trace, either scan or store, and computes a time domain representation.

The data that is processed may be any of the reflection, transmission or the reverse port versions. When enabled, additional display options are available for the left and right axes. A typical trace showing the reflection port time domain is as follows



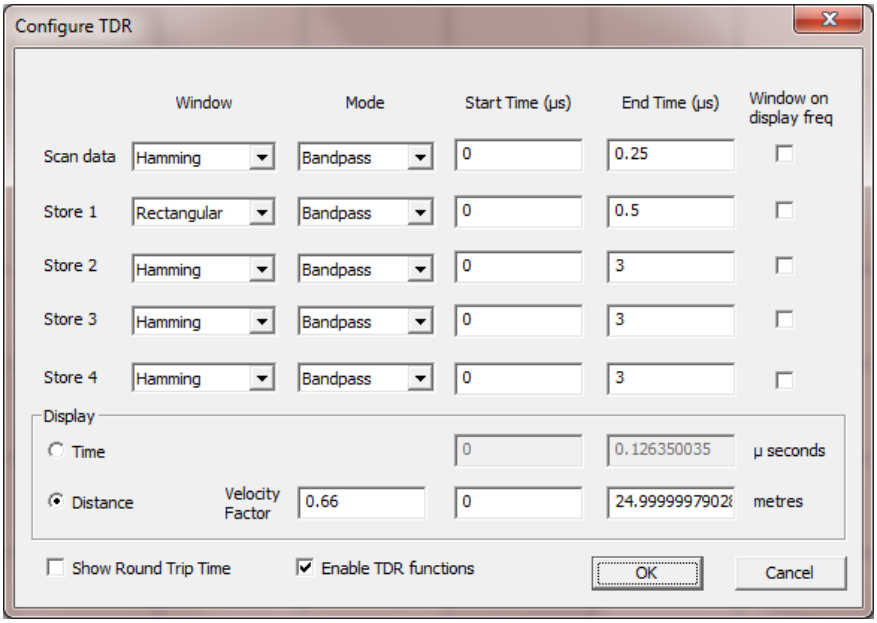
Markers display with time instead of frequency when attached to a time domain trace. In addition the status line displays time as well as frequency for the cursor position.

The time axis may be moved / scaled in the same way as the frequency axis by holding down the CTRL key when dragging with the mouse.

The program is capable of computing a time domain representation from complex frequency domain data as described by Agilent in the application note “Agilent Time Domain Analysis Using a Network Analyzer”, Application Note 1287-12.

The program implements the “Chirp Z-transform” to perform the calculation using either low pass or bandpass modes.

Here is the configuration dialog:



For each of the data sets, Scan and Stores 1-4, the configuration may be separately set. The Window provides a drop down to select the window type. Mode permits selection of Lowpass or Bandpass. The computation and display times are separately set. Currently only low pass impulse response is supported not step response.

The Bandpass mode is the most general purpose mode, permitting calculation over an arbitrary frequency scan and number of points, save that it must be a linear scan and at least 100 points.

The Lowpass mode requires a scan where the steps are linearly spaced such that the first point frequency is equal to the difference between the first two points. If this is not the case, then for a scan the program will switch back to bandpass mode and for a store will switch off time domain plots for that store.

Look back at the configuration dialog. The tick option against each trace permits the frequency range for the transform, which is normally all the data points in the scan from start to stop frequency, to be windowed on the current display frequency. Hence you may zoom in on a part of the scan and transform it. The zoom must contain at least 100 points but the output time domain scan will always contain the full number of points as the full frequency scan. The minimum frequency used will be the highest of the display start and the scan start frequencies and the maximum will be the lowest frequencies of the scan and display end frequencies.

The Display items in the dialog control how the trace is displayed. Firstly, the axis, markers and status bar may be set to display values as time or as “distance”. If “distance” is selected, then the velocity factor for the transmission line needs to be defined. Finally, the start and end times for the display may be set.

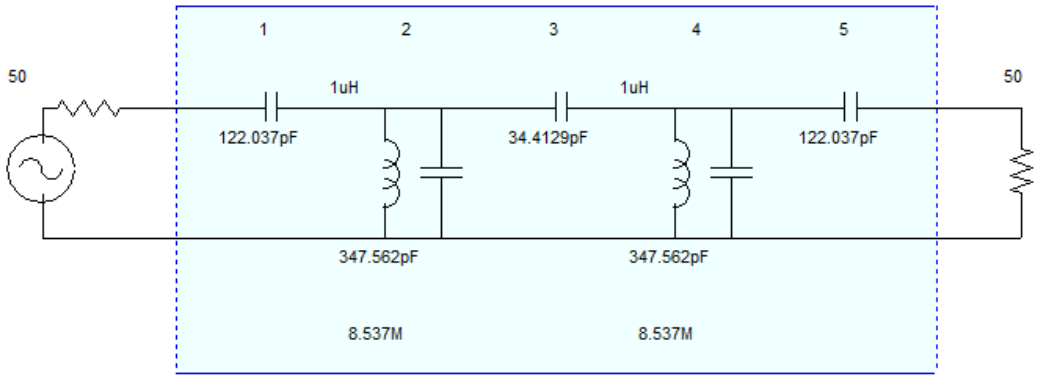
In time mode, the displayed values are in µs. In distance, values are shown in metres. The distance is computed as the product of the time, the velocity factor and the speed of light.

There is a tick box at the bottom left of the dialog box labelled “Show Round Trip Times”. Program builds prior to version 90 displayed the time as double the actual time of propagation along a transmission time. This is misleading, and it has been changed in build 90. For backwards compatibility, if this is needed, tick this box to display data in the previous form, however expect distance measurements to be double (and incorrect) in this case.

Below is a worked example of filter tuning using time domain mode kindly furnished by Roderick Wall VK3YC.

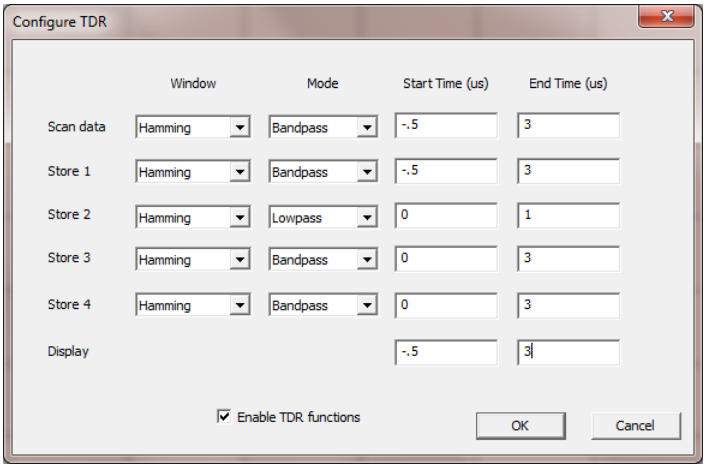
23.1 Using Time Domain to tune variable coupling BPFs

A Template is required to tune variable coupling filters. The Template can be generated from a Gold Standard Engineering filter or simulated in software. This example uses Jim Tonne's Elsie Filter design software to design a BPF and to generate an s2p Touchstone s-parameter file. The s-parameter file is loaded into a store in myVNA to be used as a Template to tune the filter. The example 40 metre variable coupling BPF circuit is shown below. Trim capacitors were used for the coupling capacitors. Adjustable inductors were used for the inductors.

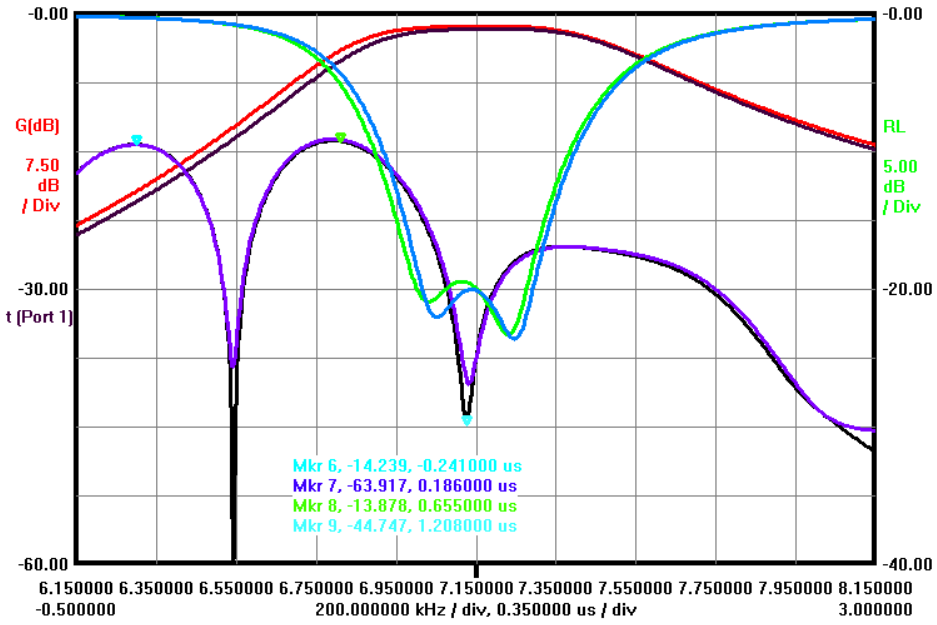


Use Elsie to design the filter. Set Inductor/Capacitor Q values to suitable values. Set the centre frequency to 7.15MHz, frequency span to 2MHz and sweep steps to 500. The centre frequency must be set to the resonant frequency that the resonators are to be tuned to. If you don't then you will be tuning the BPF to the wrong frequency. Set the span to two to five times the bandwidth of the BPF.

Generate an s2p s-parameter Template file. Load the s2p Template file into myVNA Store 1. When asked, select yes to make the Frequency range and steps the same as the loaded s2p Template file. Under “Trace Calculations” select for both Scan Data and Store 1 “TDR Configurations” and set: Window = Hamming, Mode = Bandpass, Start Time = 0.5us, End Time = 3us and select “Enable TDR functions” as shown below. Set the display time to match. Click “OK”.



Under “Display Left” Select “G[dB]”, press the Control key and select “TDR-S11”. Under “Display Right” select “RL” (Return loss) as shown below. Set the display “Vertical Axis” for suitable scales. Note, the following also shows the traces for the DUT BPF that has been tuned.



The traces are as follows:

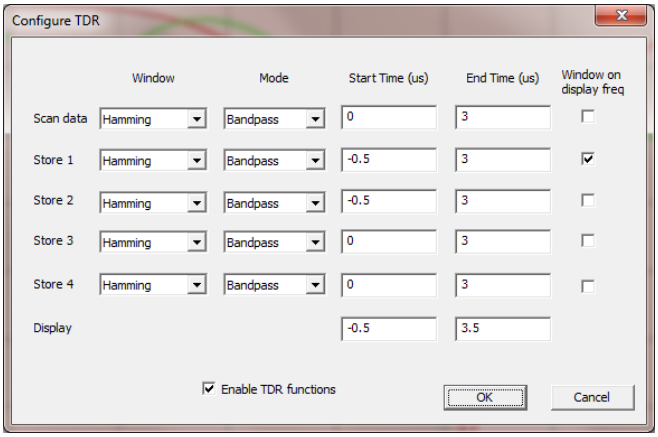
Purple t-S11 Time Domain trace = Elsie Template.
Black t-S11Time Domain trace = DUT BPF.
Dark Green Return Loss trace = Elsie Template.
Light Green Return Loss trace = DUT BPF.
Black G[dB] Insertion Loss = Elsie Template.
Red G[dB] Insertion Loss = DUT BPF.
Click “single scan” to scan the DUT BPF.

Your Time Domain trace won't have the deep dips shown above because your DUT BPF has not been adjusted. The two Time Domain dips (Time Markers 7, 9) are the return loss from each resonator displayed on a time scale. The three humps (Time

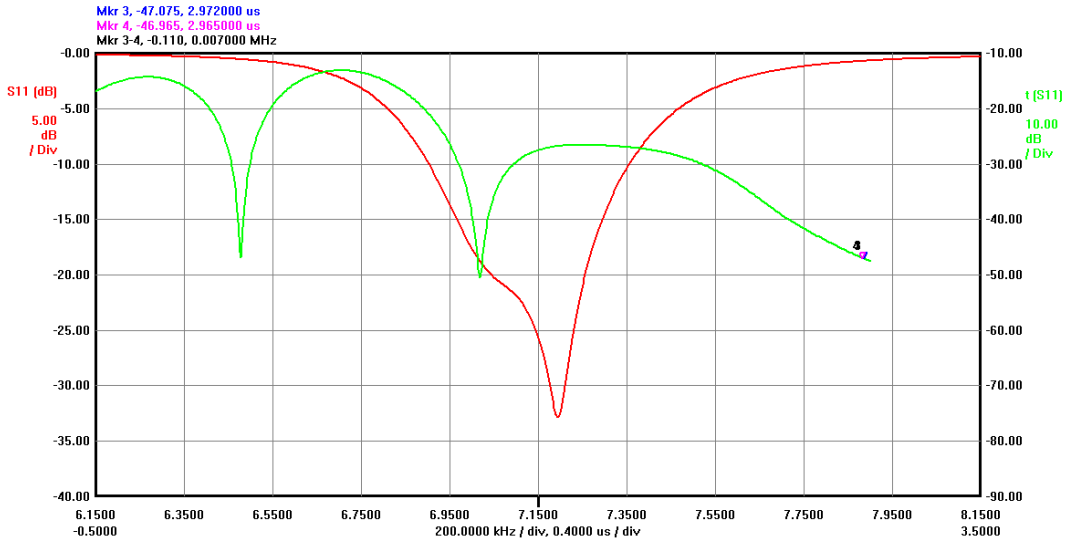
Markers 6 &8) are the three coupling capacitors. Refer to Agilent's application notes AN 1287-8 and AN 1287-10 for the procedure to adjust the filter. Click "Start Scan" to sweep and adjust the BPF.

Before assembling the BPF you may want to adjust the components to the values shown in the circuit, this will make it easier to adjust the BPF.

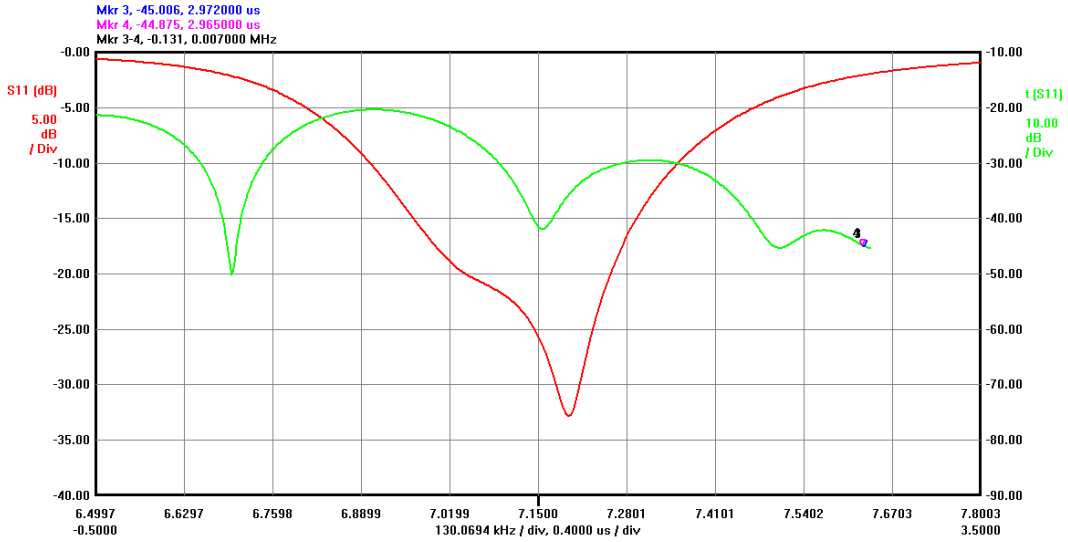
Adjusting the centre frequency of the transform allows you to determine the frequency the resonators are tuned to. Adjust the centre frequency for the deepest dip and the centre frequency is the resonant frequency. Refer to Agilent application notes for more information. To do this, in this example with stored data. Look again at the TDR configuration dialog.



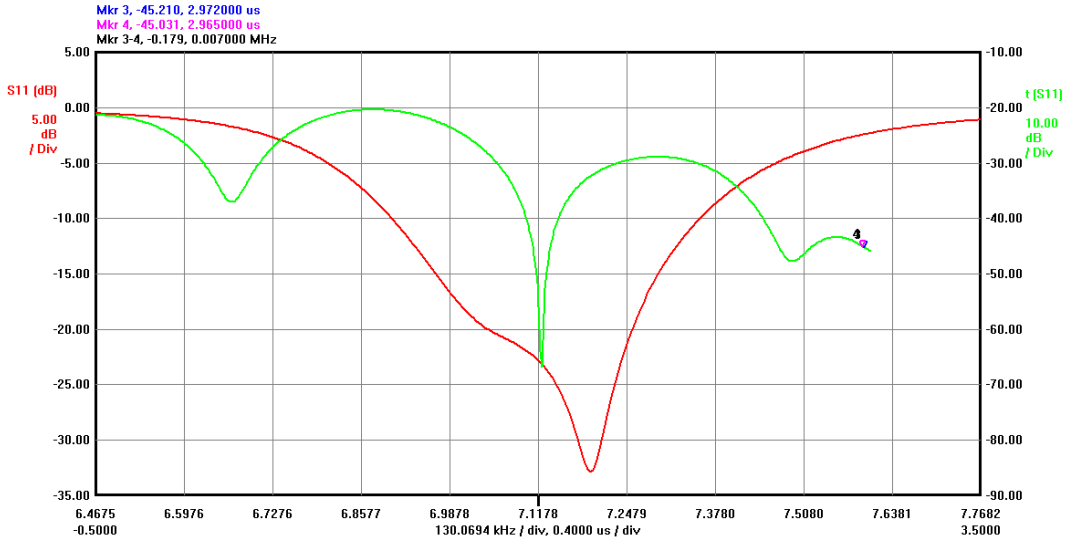
Note the "Window on Display freq" option. When this is ticked, the frequency range used for the transform is clipped to the range of frequencies shown in the display. So, given this display



Zoom in slightly on the frequency axis by putting the cursor on the frequency axis, press and hold the left mouse button and drag the cursor left to zoom in slightly like this



Now put the cursor in the main display grid, press the left mouse button and hold, then drag left or right and observe the trace, for example thus



The depth of the nulls will change as the trace is dragged. In this case the centre frequency is 7.1178 MHz for the second resonator.

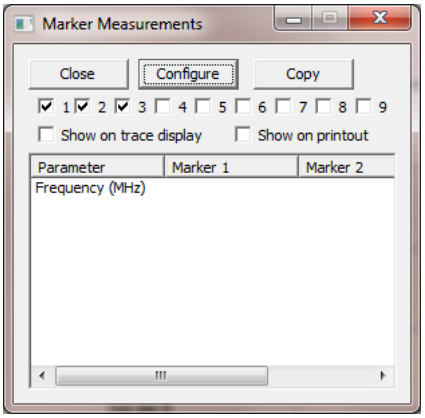
The amount of data that may be used in such a zoom is limited. The data will be clipped to the display frequencies and the start & stop frequencies of the data, and the amount of data must exceed a (fairly arbitrary) 100 points. The output time domain data will always be the same number of data points as the trace itself – in this case 501 points. Also if you zoom in too much you get some strange results.

24 Marker Measurements

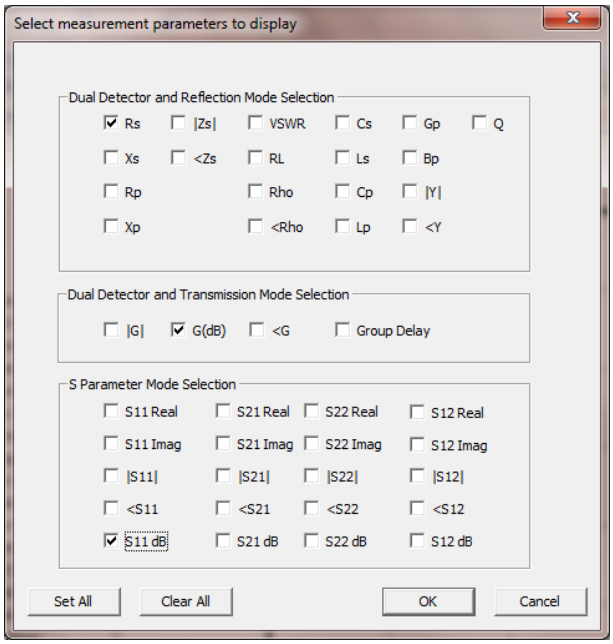
To extend the functionality of markers, a separate marker measurements dialog box may be opened to display extra data. This chapter describes its operation.

Under “Trace Calculation” is an option “Show Marker Measurements”. This opens a dialog box that provides a detailed view of a range of parameters at each of the markers (1-6).

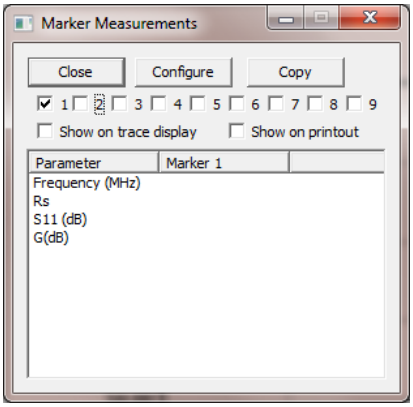
When opened (it may also be opened from the toolbar or from a right click / popup menu in the main view) a dialog box appears like this



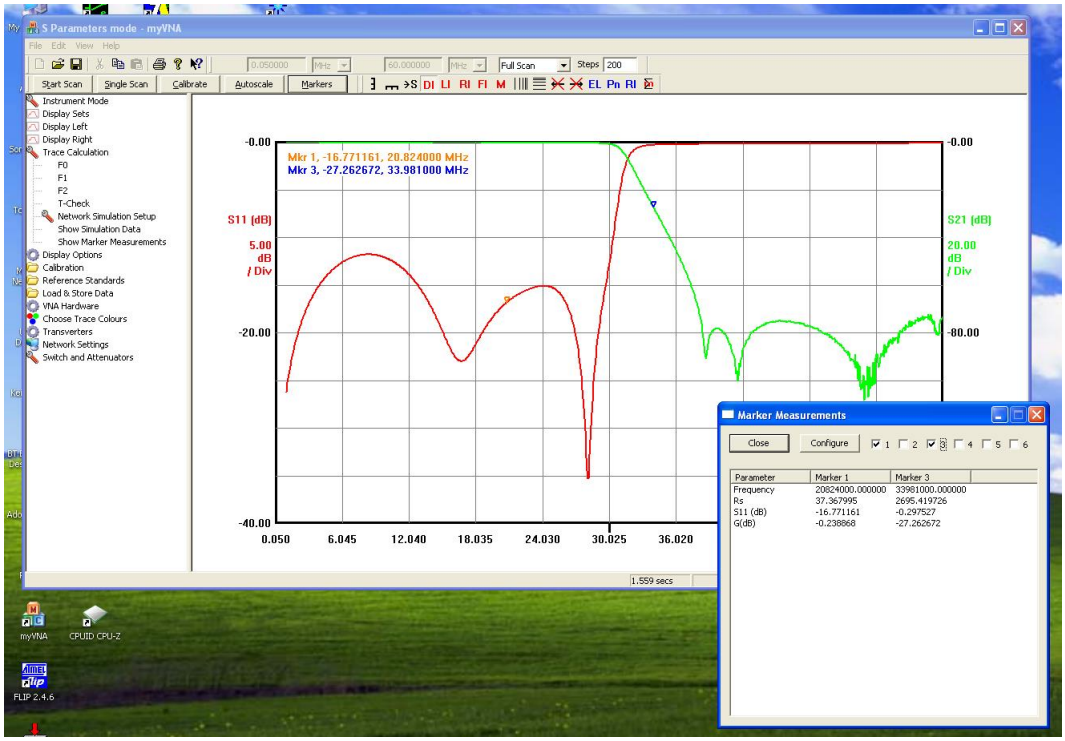
This, when opened, will always appear on top of the main window. It may be minimised, or it may be closed and re-opened. The settings will still be present (they are also saved on program exit / restart). The “Configure” button determines which parameters are to be displayed. It brings up a dialog box thus:


Select measurement parameters to display
Dual Detector and Reflection Mode Selection
Rs |Zs| VSWR Cs Gp Q
Xs <Zs RL Ls Bp
Rp Rho Cp |Y|
Xp <Rho Lp <Y
Dual Detector and Transmission Mode Selection
|G| G(dB) <G Group Delay
S Parameter Mode Selection
S11 Real S21 Real S22 Real S12 Real
S11 Imag S21 Imag S22 Imag S12 Imag
|S11| |S21| |S22| |S12|
<S11 <S21 <S22 <S12
S11 dB S21 dB S22 dB S12 dB
Set All
Clear All
OK
Cancel

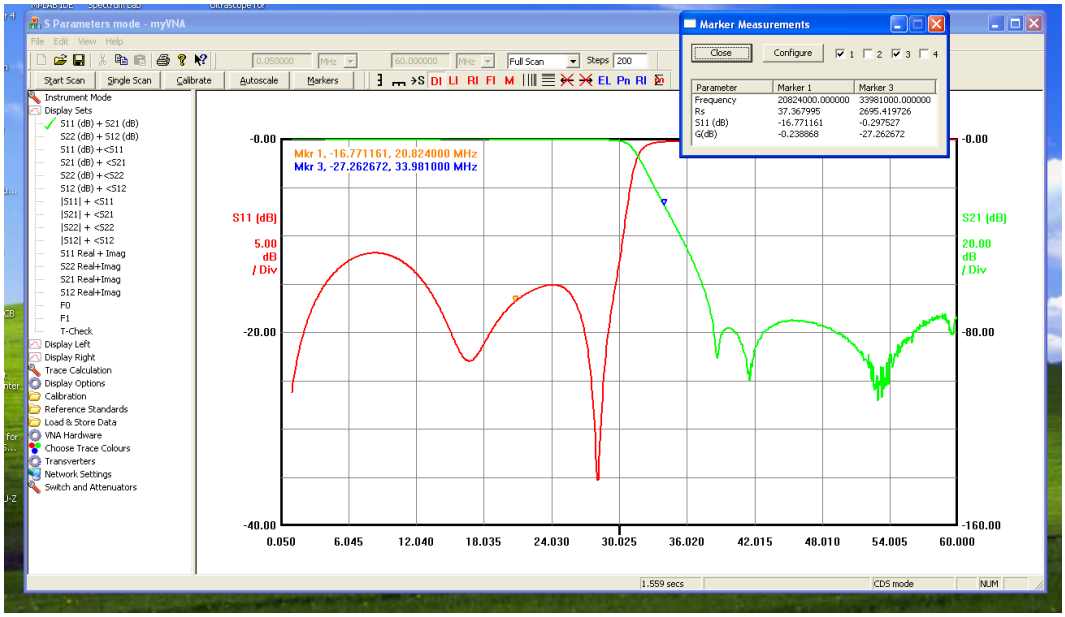
Tick the parameters desired; for example ticking “Rs”, “G(dB) and “S11dB” followed by “OK” changes the marker measurements window to this



For each marker selected, those parameters will be shown. To select the appropriate markers, tick the option 1 to 6 at the top. Selecting for example ‘1’ and ‘3’ will not immediately display any information, but when the markers are showing valid data then the parameters listed will be enumerated for the markers selected, for example as shown next.

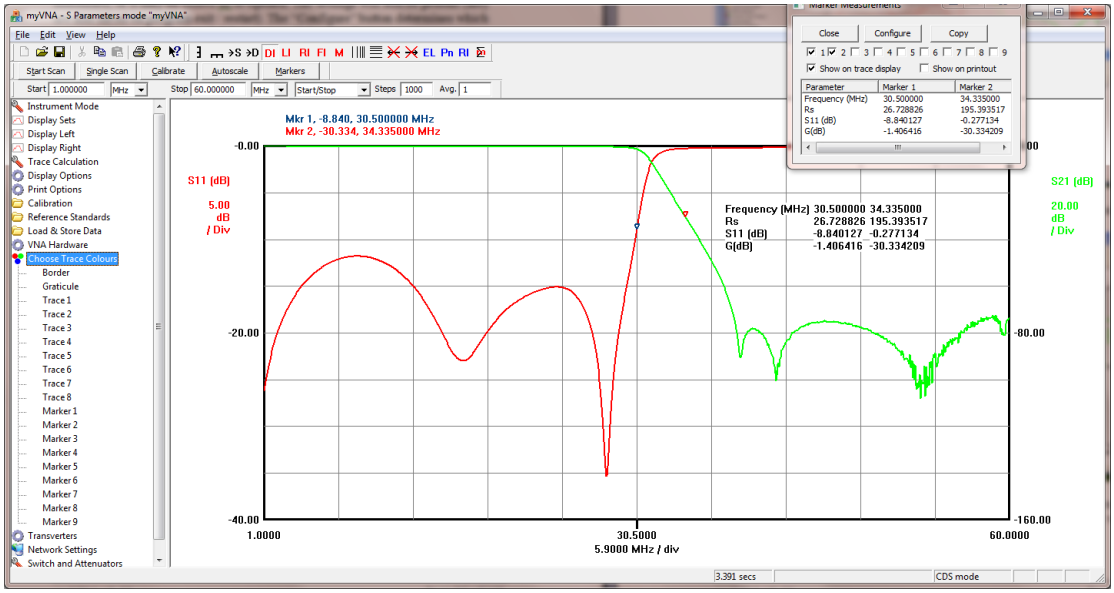


The Marker Measurements box may be moved, minimised or resized as desired



A word of warning. The markers will interpolate, for example for a zero crossing marker. The Marker Measurements will not. Also displaying Group Delay with a marker at the left hand end of the scan will give rubbish results.

Now, look again at the Marker Measurements dialog box. Underneath the controls for each of the markers, are two other options. One causes the marker measurements to be shown overlaid on the main trace display, the other causes the marker measurements to be included on the printout. For these functions to work, you must keep the marker measurements window open, but it can be minimised to the taskbar.

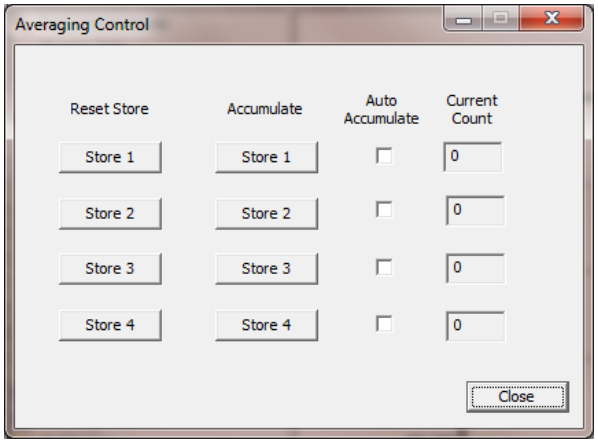


The location of the measurements results may be dragged around on the screen to position them in any desired location.

25 Trace Averaging

As well as averaging multiple readings at each point, it is sometimes useful to average a number of traces. This chapter shows how that is done.

Available from the secondary toolbar or the treeview (under Trace Calculation) is the option to average traces. This opens a dialog box as shown



This dialog box may be left open, minimised or closed & reopened as desired. It permits traces to be averaged provided that each trace is the same (frequency, number of points).

The basic operation is as follows. Traces that result from a scan may be manually or automatically averaged with other traces in one of the stores. By way of example, clicking the “Store 1” button under Reset Store results in the current scan data being stored in Store 1 and the Current Count being set to 1. Each time the “Store 1” button under “Accumulate” is clicked, the current scan data is added to a running average in Store 1 and the count incremented.

It is also possible to automatically average each trace. Checking one of the “Auto Accumulate” buttons then performing a scan causes the scan to be added to the average at the end of the scan.

There is one point to be careful of. The Reset and Accumulate buttons may be clicked at any time, for example during a scan. This is useful to reset the running average during a scan, but it takes a snapshot of the current trace data so if it is not valid, you get an invalid average.

26 Signal Generator

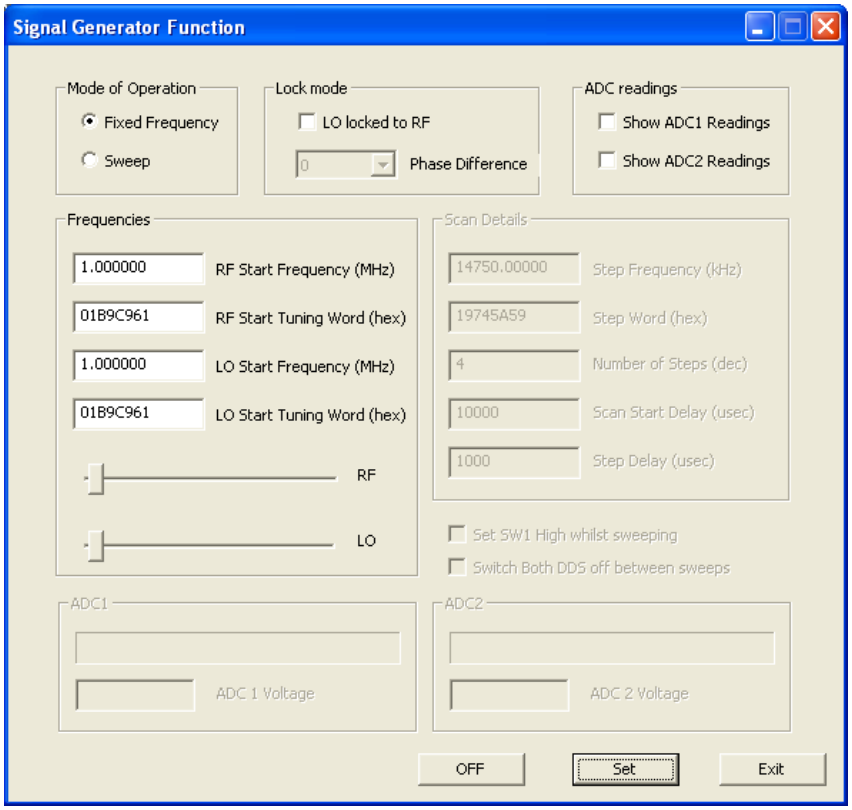
The signal generator provides a fixed or sweepable signal generator. All functions work for USB but there is no sweep option for the parallel interface. This chapter describes its operation.

Start by setting the mode (top right of the dialog box) – sweep or single frequency. On the right of that, you may select to lock the LO to the RF frequency. When you do that the controls to set LO frequency are greyed out. This is a general principle, as you change the mode, boxes will be greyed out or made visible according to their relevance.

Having locked the LO and RF you may then set the phase – there is a drop down list with all possible phase settings.

Set the scan details either by frequency or DDS tuning word. If you update a frequency, the DDS tuning word is automatically updated. Similarly, edit the tuning word and the frequency changes. Note that frequencies honour the current transverter setting – the frequencies must be valid for the band. You may also change the frequency by dragging the slider bar left or right for RF or LO. Alternatively, if you select a slider bar by clicking on it, the cursor left and right buttons will give finer control. There are 32767 points on the sliders, so with a 60 MHz range on the VNA, each step is just under 2 KHz apart.

The option to set SW1, if ticked for a sweep (it is only valid in a sweep) causes the SW1 output to go high when the scan is valid and low when resetting for the next sweep. Similarly if you select to power down the DDSs between sweeps the RF and LO outputs will switch off between sweeps.



The ‘OFF’ button is greyed out until the frequency is set. Whilst the Power Down DDS hardware option will switch off the signal generator when you exit the function, this button may be used to explicitly switch off the signal.

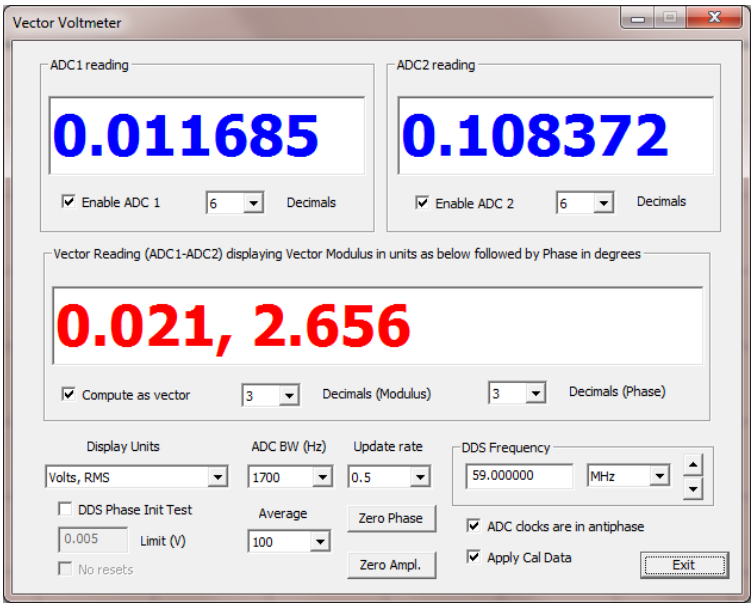
27 Vector Voltmeter

The Vector Voltmeter uses both ADCs and a modified configuration to measure the amplitudes and relative phase of two RF signals. It may also be used to test the DDS synchronisation. This chapter describes its operation.

The method here is attributable to Ivan Makarov VE3IVM. To use the VVM mode, connect one signal to each of the two detector RF inputs. The VVM is a narrow band mode, so it is important that the frequency be set to the same as the signal being measured.

There are two components to the VVM, a calibration screen and a VVM operational screen. The VVM will work without calibration, but the levels will be inaccurate at high frequency, tailing off by a few dB.

Looking first at the VVM operational screen, which is accessed from the “Instrument Mode” node on the leftview, titled “Vector Voltmeter”. The screen will look something like this



The display shows, at the top, the current ADC readings for each ADC. Each may be switched on or off and the number of decimal points shown may be changed. Below that, in the middle of the screen is the difference between ADC1 and ADC2 in amplitude and phase. The format of this data is controlled by the settings lower down on the dialog and may be either a simple value difference, a vector computation, or a ratio.

Bottom left, there is a drop down list that selects the units for the display. This may be in volts or millivolts, either RMS or peak-to-peak or in dB / dBm. It also permits a simple ratio to be displayed or a ratio multiplied by the system reference impedance (typically 50 ohms).

From left to right at the bottom, the ADC mode is selected, the speed of update of the screen display is set, the number of traces that are averaged is set, the relative phase and magnitude / ration may be set to zero, the frequency of the meter is determined and calibration data applied.

The ADC mode is set from a drop down list; and the settings are the same as those of the ADC modes, but here are displayed as the bandwidth that is applied as a result of the mode choice.

The meter will display a rolling average of the last N readings; a drop down list allows selection of the value N.

The meter runs continuously in the background, but if it were to update the screen each reading it would flicker too much to be of use. Therefore it updates the screen at a rate selected by the user in seconds.

The meter reads the amplitude and phase of signals at a given frequency; that frequency is set by the controls at the far right.

The "zero phase" button may be used to zero the current phase difference; useful in correcting for cable differences.

The "Zero Ampl." Button may be used to zero the current difference (set to zero for difference calculations, set to 1.0 for ratios).

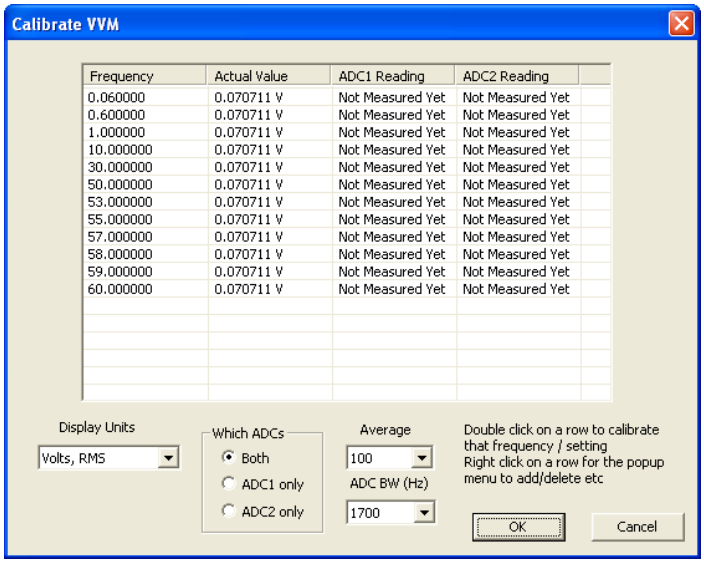
It is usually the case that the two local oscillator DDS signals are 180 degrees out of phase. To provide an easy correction for this, a 180 degree offset may be applied with the tickbox.

Finally, the current calibration data may be applied or not to the readings. This data corrects for amplitude variations and mainly affects high frequency response beyond 55 MHz.

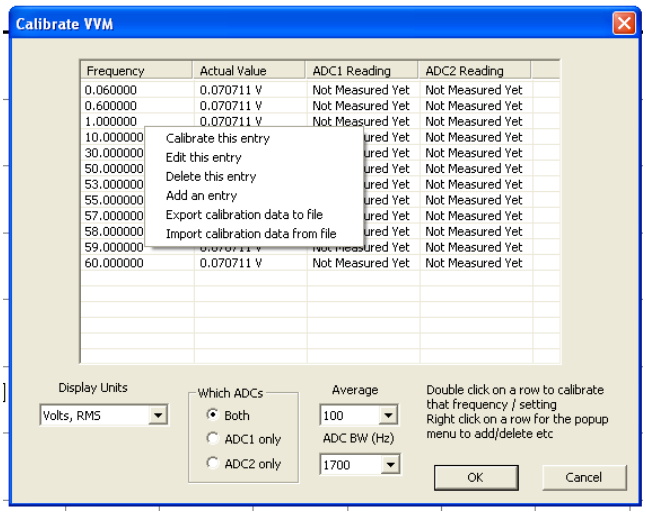
> *A specific control to note is the "Compute as Vector" checkbox just below the vector reading. This conditions the basic calculation method. There is no difference for the ratio measurements; it affects the voltage difference computations.*
>
> *When unchecked, the voltage differences are computed as a simple subtraction of the amplitudes of the signals and of the phases. When checked however a vector calculation is performed.*

So, to try the meter out, connect both ADC inputs to a T piece and apply a signal. Set the frequency to be the same as that frequency and observe the display. The relative amplitudes should be about the same and the phase somewhere near zero. Now put a couple of metres of coax in one leg of the T piece. The relative phase and level differences should be shown.

There will be some error in the amplitudes. To correct for this, there is a calibration function under "calibration" in the leftview. It brings up a dialog box like this:


Calibrate VVM
Frequency
Actual Value
ADC1 Reading
ADC2 Reading
0.060000 0.070711 V Not Measured Yet Not Measured Yet
0.600000 0.070711 V Not Measured Yet Not Measured Yet
1.000000 0.070711 V Not Measured Yet Not Measured Yet
10.000000 0.070711 V Not Measured Yet Not Measured Yet
30.000000 0.070711 V Not Measured Yet Not Measured Yet
50.000000 0.070711 V Not Measured Yet Not Measured Yet
53.000000 0.070711 V Not Measured Yet Not Measured Yet
55.000000 0.070711 V Not Measured Yet Not Measured Yet
57.000000 0.070711 V Not Measured Yet Not Measured Yet
58.000000 0.070711 V Not Measured Yet Not Measured Yet
59.000000 0.070711 V Not Measured Yet Not Measured Yet
60.000000 0.070711 V Not Measured Yet Not Measured Yet
Display Units
Volts, RMS
Which ADCs
Both
ADC1 only
ADC2 only
Average
100
ADC BW (Hz)
1700
Double click on a row to calibrate that frequency / setting
Right click on a row for the popup menu to add/delete etc
OK
Cancel

This is an “empty” calibration data set. It permits correction for amplitude variation for each ADC. For each row, which corresponds to a specific frequency, the difference between an actual signal and that read is determined and used to correct the displayed values. Right click on the display and a popup menu appears



This menu is used to manage the calibration data. If the mouse was over a specific row in the table, it allows a calibration to be performed for that row, or the data in that row to be edited or the row deleted. New rows may also be added, and the complete set of data may be exported to file or imported from file.

To calibrate proceed as follows

1. review the table and add or delete rows as needed
2. apply a known signal at one of the frequencies in the table at the level shown under “actual” to one of the ADCs
3. select the ADC to which the signal is being applied
4. Double click on the row, or right click and select ‘calibrate’

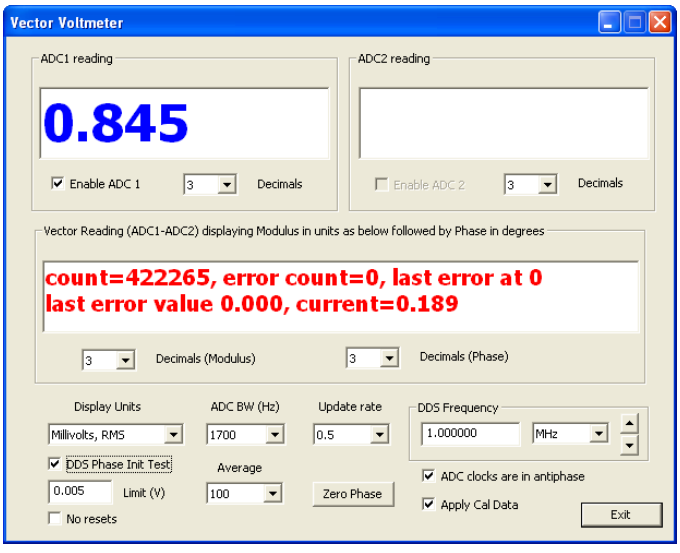
5. Check that the correct row under the ADC(s) selected now shows a reading. It should be similar to that applied or else there is a fault somewhere
6. Repeat for all points and both ADCs

The calibration data does not need to be explicitly saved and loaded but it is useful to do so. Current calibration data is saved in the registry on exit and automatically reloaded on startup.

Click 'cancel' to exit discarding all changes made to the current calibration data. Or click 'ok' to exit and keep it

27.1 DDS Reset Test

The Vector Voltmeter also serves as a placeholder for a diagnostic tool; bottom left there is a greyed out box and a tickbox labelled "DDS Phase Init Test" . This is used to check on the phase stability of hardware following a DDS reset & load. To use it, connect a short cable from RF output to RF detector input. You should get a good high voltage of about 360mV rms. Now tick the "DDS Phase Init Test" box. The display will change to something like the following. The voltage shown will (or should) drop significantly as the VVM display will now show not the modulus of the signal but the amplitude of the quadrature component. This, if the relative signal delays between RF and LO are about the same, be close to zero but expect some deviation. Before using this however, select a frequency of 1 MHz with the 1700 Hz ADC bandwidth. Also note that the rate will depend on the phase and step delays. The VVM will remember the first reading and then compare all subsequent readings to it. If it finds one that differs by more than the amount in the Limit box, it will flag that as an error.



In the above example, the test has been running for 422,265 measurements without an error. What it is doing is to make sure that when the DDSes are reset, they faithfully come up in the correct phase synchronisation.

Now a word of warning; if the voltage before you tick the DDS Phase Init Test box is NOT above, say, 300mV or the voltage when ticked is not below about 10mV then STOP; there is something wrong. Try again and if you still don't get the right values, there is probably a serious fault somewhere.

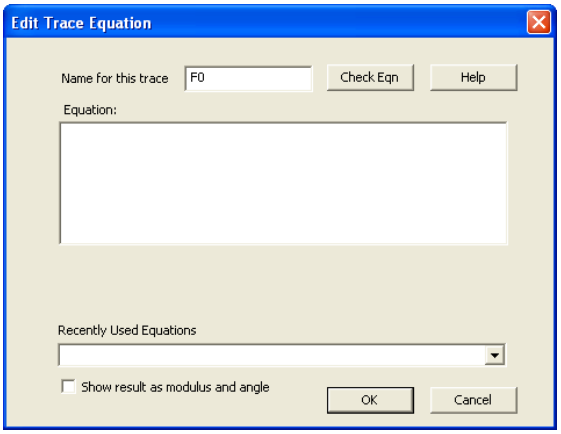
The second (simpler) test omits the reset of the DDSes between each test so merely checks to see that the ADC readings are stable.

Give it a good run. You should get to a million without errors. It is also a good test of the combination of PC, operating system, interface and VNA.

28 User Defined Display Functions

User defined functions represent a powerful tool for calculations. Scan, stored or simulation data may be combined in user defined equations that may then be displayed. This chapter describes how it is used.

There is a Treeview entry on the left called “Trace Calculation”. Open it up and 4 entries are shown, labelled initially “F1”, “F2”, “F3” and “T-Check”. Clicking on one of these entries opens up a dialog box as shown below.



This is an equation editor that allows custom display traces to be created. The name, F0, may be changed, and this name will then be used in the display choices (left, right or sets) and in the labelling of the axes; but keep the name short.

The equation editor manages a complex number calculator that takes input from scan or store data and creates data that may be displayed graphically or in the report view.

An equation might be something simple like this:

```
-1e12 / ( w * scan(xp) )
```

or it may be much more complicated. The above example calculates the parallel capacitance from the scan data in the same way that the built in function for cp does. Specifically, it multiplies the real value for xp by 2*pi*f and divides it into -1e12 to give cp in picofarads.

The last entry is by default the T-Check algorithm from R&S courtesy of Paul. The formula is as follows:

```
abs(scan(s11)*conj(scan(s21))+scan(s12)*conj(scan(s22)))/(sqrt((1-
abs(scan(s11))^2-abs(scan(s12))^2)*(1-abs(scan(s21))^2-abs(scan(s22))^2)))
```

What the program does is to take the equation, compute a complex result for each scan data point and provide the results to the display routines. The display routines will show the result as either real & imaginary or modulus and angle (in degrees). Whenever a valid equation is present in one of the four options, two new entries will be present in the left and right data choices for each valid equation (real and

imaginary or modulus and angle) and one new entry will be present in each display set. This may be selected and will then display the results in graphical or report views.

The equation may comprise data sources, functions, values and constants.

When you exit from the function with "OK", the program checks the syntax of the equation and compiles it. If it finds an error it will not exit but will display an error message. You may also check the syntax without exiting by using the "check" button. Checking is limited; the best way to view it is GIGO – give it a garbage equation and you will get garbage results.

The dialog also maintains a list of the last 10 functions used; at any time you may select one from the drop down list and it will replace the current function with the one selected.

28.1 Data Sources

The data sources are the current scan data or any of the four saved data traces. These are accessed using the functions scan() and store(). In each case the function takes one parameter chosen from the following choices, and in the case of the store() function it also takes the store number. Examples therefore might be

scan(s11) or
store(1, rho)

The first example retrieves the current scan s11 complex data. The second retrieves the complex reflection coefficient data from store 1. The first parameter for the store function must be a value from 1 to 4. The second should be one of the parameters shown below; in fact these are just pre-defined constants; you can see the value by entering a function of just the parameter name such as "xs" and seeing the effect in report view.

It is important to note that the stored data must exist to be used and that the number of data points in the store MUST be the same as the number in the current scan. The program will try to prevent you selecting an equation that requires a store if the store does not contain data of the same number of points but try hard and you can probably get round this.

Parameter	Load the following
zs	Series impedance
rho	reflection coefficient Rho
y	Complex admittance
s11	S11
s21	S21
s22	S22
s12	S12
gain	Transmission gain

In each of the above, the complex value for the parameter is loaded. This is used directly in calculations as a complex quantity.

It is also possible to load individual calculated results as shown in the table below. Note however that in each case these are loaded as a real value into the calculator, or more precisely as a complex value with a zero imaginary part.

Parameter	Load the following
rs	Series Resistance
xs	Series Reactance
rp	Parallel Resistance
xp	Parallel Reactance
modzs	Modulus of series impedance
angzs	Phase of series impedance in degrees
vswr	VSWR
rl	Return Loss
modrho	Modulus of reflection coefficient Rho
angrho	Phase of reflection coefficient Rho
q	Q
cs	Series capacitance
ls	Series inductance
cp	Parallel capacitance
lp	Parallel inductance
gp	Real component of admittance
bp	Imaginary component of admittance
mody	Modulus of admittance
angy	Phase of admittance in degrees
rhore	Real component of reflection coefficient Rho
rhoim	Imaginary component of reflection coefficient Rho
s11re	Real component of S11
s11im	Imaginary component of S11
s11mod	Modulus of S11
s11ang	Phase of S11 in degrees
s11rl	S11 return loss
s21mod	Modulus of S21
s21db	Modulus of S21 in dB
s21ang	Phase of S21 in degrees
s21re	Real component of S21
s21im	Imaginary component of S21
s22re	Real component of S22
s22im	Imaginary component of S22
s22mod	Modulus of S22
s22ang	Phase of S22 in degrees
s22rl	S22return loss
s12mod	Modulus of S12
s12db	Modulus of S12 in dB
s12ang	Phase of S12in degrees
s12re	Real component of S12
s12im	Imaginary component of S12

modgain	Modulus of transmission gain/loss
gaindb	Transmission gain in dB
anggain	Phase of transmission gain/loss in degrees
gainre	Real component of gain
gainim	Imaginary component of gain

So an equation that loaded S11 from the scan data would be

scan(s11)

And loading the modulus of the transmission gain from data store 1 would be

store(1, modgain)

It is important to remember this; if you put into a formula of "s11" you will not get the s11 data – you will just get an "internal program" constant. You must use "scan(s11)" or one of the store functions.

The editor is not case sensitive so scan(s11) gives the same result as ScAn(S11)

It is also possible to include in a calculation other data points from the scan. Two functions support this:

iscan(index_offset, what)
istore(store, index_offset, what)

These "indexed" versions (hence the names with an 'i') allow an offset to be added. An index_offset of 0 gives the same result as scan() or store(). A value of 1 gives the data point after the current point, -2 gives the point two before the current point etc. Hence by way of example, the equation "(iscan(-1, s11)+scan(s11)+iscan(1,s11))/3" would give an average of the current point and the ones before and after it. Another way to achieve the same effect is to use the avgscan() or avgstore() functions which provide a simple way to average a range of points about the current point.

avgscan(range, what)
avgstore(store, range, what)

A range of, for example, 2, will return the average of the current point, the two before and the two after.
If a point is accessed before the first point in the scan or after the last point for any of these four offset functions, then the first or last respectively will be returned to avoid strange effects at the start or end of a plot.

It is possible also to access raw scan data – this is the complex ADC data before any processing is applied. Specifically this is before OSL or n-term corrections and is therefore a simple indication of the ADC voltages read. The data are accessed thus:

rawscan(index)

where index is as follows:

0 Forward transmission
1 Forward reflection
2 Reverse transmission

3 Reverse reflection

Only raw scan data may be accessed in this way – not stored data.
One possible use for this data is to perform swept vector voltmeter measurements.

As a final twist, it is also possible to load the results of a calculation into another calculation, so for example

scan(calc1)

would load the complex result of the first calculation (initially named "F1"). Be careful however, recursion is not allowed and calculation is sequential, so loading calc1 into F2 makes sense but loading calc2 into F1 does not.

28.2 Functions

There are a small number of built in functions that may be used to build an equation. The following operate on complex data values. All angles in these equations are in radians.

Function	Description
cos(x)	Calculate cosine of x (complex) giving complex result
sin(x)	Calculate sine of x (complex) giving complex result
tan(x)	Calculate tangent of x (complex) giving complex result
cosh(x)	Calculate hyperbolic cosine of x (complex) giving complex result
sinh(x)	Calculate hyperbolic sine of x (complex) giving complex result
tanh(x)	Calculate hyperbolic tangent of x (complex) giving complex result
acos(x)	Calculate arccos of real component of x, setting the imaginary part to zero
asin(x)	Calculate arcsin of real component of x, setting the imaginary part to zero
atan(x)	Calculate arctan of real component of x, setting the imaginary part to zero
fabs(x)	Return the absolute value of the real component of x, setting the imaginary part to zero
int(x)	Return the integer value of the real component of x, setting the imaginary part to zero
exp(x)	Return the complex result of e^x
ln(x)	Return the complex result of the natural log of x
log(x)	Return the complex result of the log base 10 of x
sqrt(x)	Return the complex square root of x
real(x)	Return the real part of x setting the complex part to zero
imag(x)	Return the imaginary part of x as the real part, setting the imaginary part of the result to zero
im(x)	Return the imaginary part of x, setting the real part of the result to zero
abs(x)	Return the modulus of x
arg(x)	Return the phase of x in radians
conj(x)	Return the complex conjugate of x
norm(x)	Return the norm of the complex number x
polar(x,y)	Return the complex number that results from the real modulus of x with real phase of y in radians

db(x)	Return 20*log10(abs(x)) with the imaginary part set to zero.
+	Adds two complex numbers
-	Subtracts the two complex numbers
/	Divide two complex numbers
*	Multiply two complex numbers
^	Raise one number to the power of another

28.3 Numbers

Numbers may be entered in the form {digits}.{digits}[e{sign}{digits}]

For example

1
2.3
-3.4e-10

In addition a number may be preceded or followed by I or j to create a complex number, for example

3+j4
2.3e-10 – 3.6e3i

Note that I or j are functions; if you put just I or j in (for example “sin(something) + j” then you will get an error. Similarly sin(somethingelse)j will give an error. In the first case use “1j” or “j1” and in the second “*1j”.

28.4 Values

The following built in values are available

Name	Provides
freq	The current frequency for this calculation
w	freq * 2 * pi
pi	3.14159……..

Finally be aware that error checking by the compiler is limited.

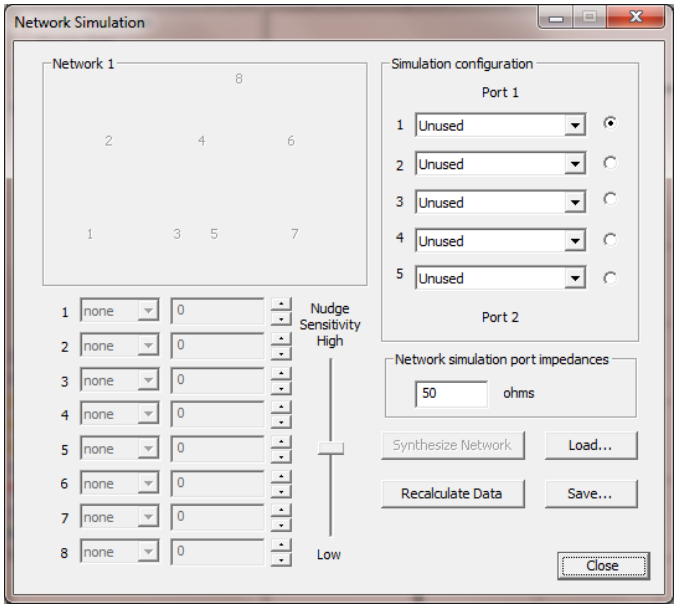
29 Network Simulation

Like the equation editor, the network simulation function allows simple networks, scan and stored data to be computed and displayed. One example of how it may be used to simulate filter matching is shown in this chapter.

Under the 'Trace Calculation' heading on the left are two options that control network simulation, titled "Network Simulation Setup" and "Show Simulation Data". The first of these opens up a dialog box that configures the simulation and the second is a toggle that displays or hides the simulation data.

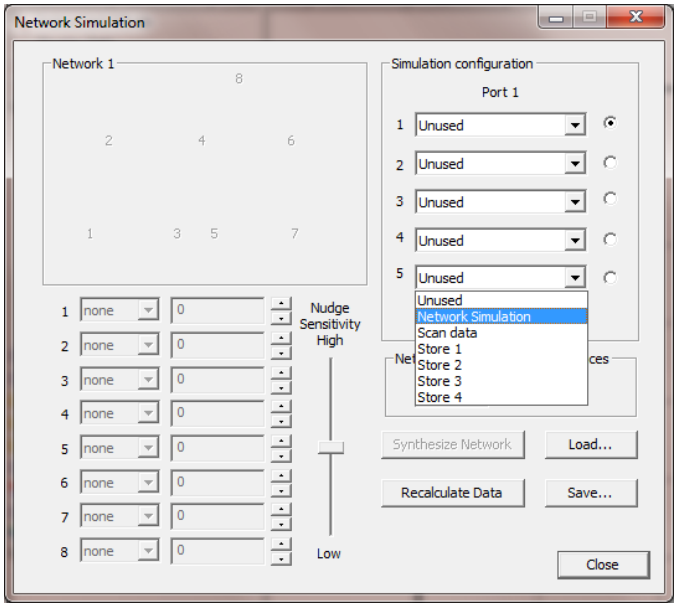
The setup screen produces a dialog box as shown below. This dialog may remain open whilst using the main myVNA application window; it does not need to be closed during use. If reopened it will display the current settings and closing it does not affect the display of simulation data.

The dialog provides a configurable simulation of five networks as listed on the right hand side. In simulation these are considered to be connected in series from 1 to 5 (top to bottom).

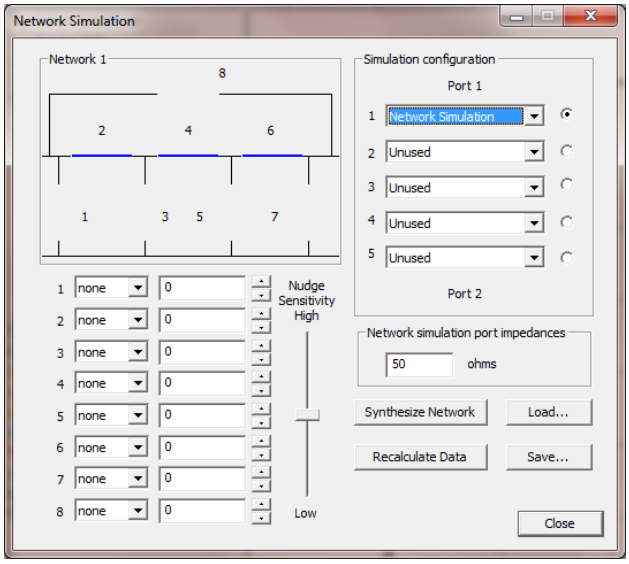


The dropdown list for each of the five networks determines its type. The following graphic shows the menu from the fifth network; all 5 display the same set of options.

The choices are to omit that network from the simulation (unused), to implement a simulation of a set of components, to use the current scan data or to use the contents of one of the data stores.



Selecting ‘network simulation’ enables the rest of the controls to the left. In each case there is a pi network of up to 7 components, each of which may be unused, a resistor, a capacitor or an inductor, plus one component in parallel with the pi network.

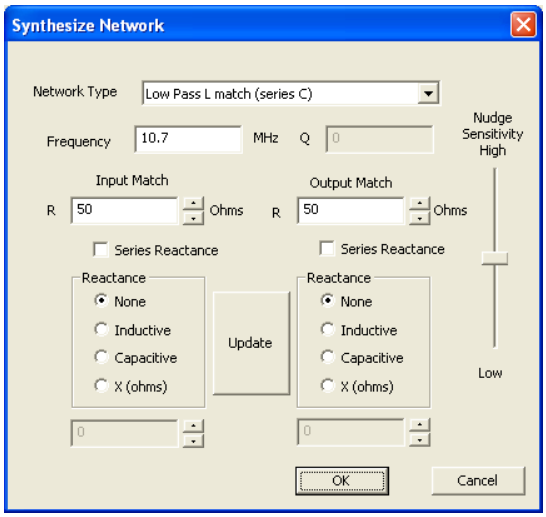


The box top left displays a graphic of the currently selected network. Each component may be selected by type from the drop down list below the graphical representation and each may be given a value. The up and down arrows on the right of each value permit the value to be nudged higher or lower in value. The slider control to the right of the nudge buttons changes the sensitivity of the nudge buttons; at the top each step is about 10% of the value; at the bottom it is about 1ppm. You may also nudge values up or down by clicking into the edit box for a component’s value and scrolling up or down using the mouse wheel.

The button lower right labelled “synthesize network” is used to generate a specific network according to design requirements. At the moment the list is limited to common matching networks.

The “Recalculate data” button needs describing. In the main, changing a network type, values or even the type of a component in a network will not cause a change to the calculated results as displayed in the main VNA window. There are however three exceptions when an update will occur. These are, respectively, when the “OK” button is pressed (and results in closure of the dialog), when the “Recalculate Data” button is pressed or when one of the component parameter up/down nudge controls is pressed.

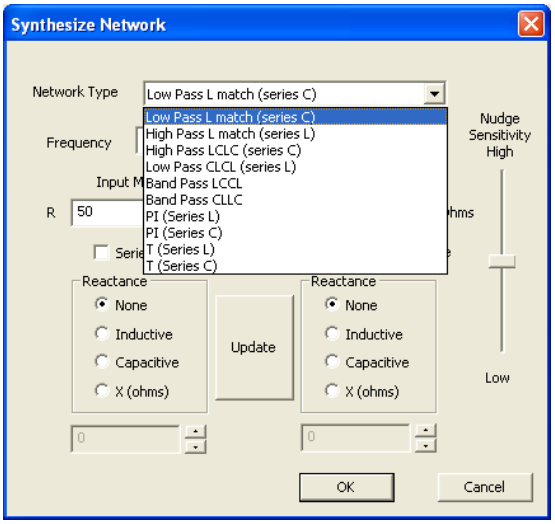
When “synthesize network” is pressed, the dialog below is opened up.



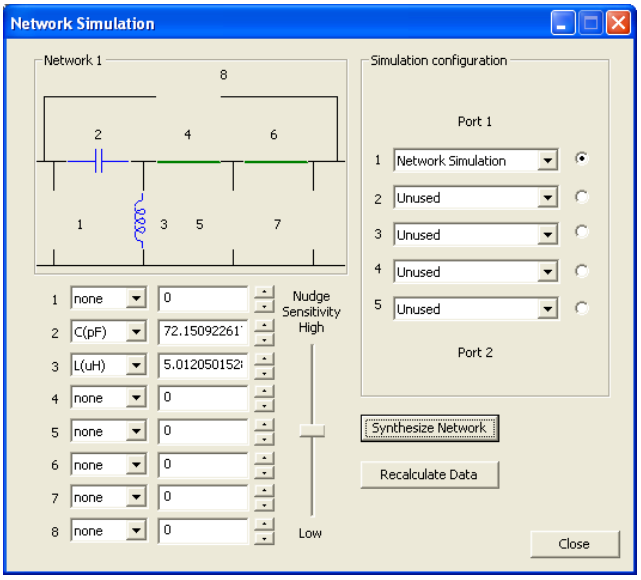
This allows selection of the network type (see below) and the setting of its design parameters. In each case a frequency and settings for input and output are needed, and depending on network type a Q value may be needed.

For each port, a target matching impedance is given as a resistive component and a reactive component. The reactive part may be specified in terms of either inductance (in µH), capacitance (pF) or simply as ohms. Also the program may be told that the resistive and reactive values given should be interpreted as in series or in parallel with the resistive part. When “OK” or “Update” are clicked the program attempts to synthesize a network of the given type. In the case of “Ok”, if it can do so, then the dialog box closes and the graphic on the network simulation dialog is updated to show the network. If it cannot (for example no match is possible for the values given) then an error message is shown and the dialog remains open. The program is quite happy to generate negative component values if you force it to but will complain if no match is possible at all.

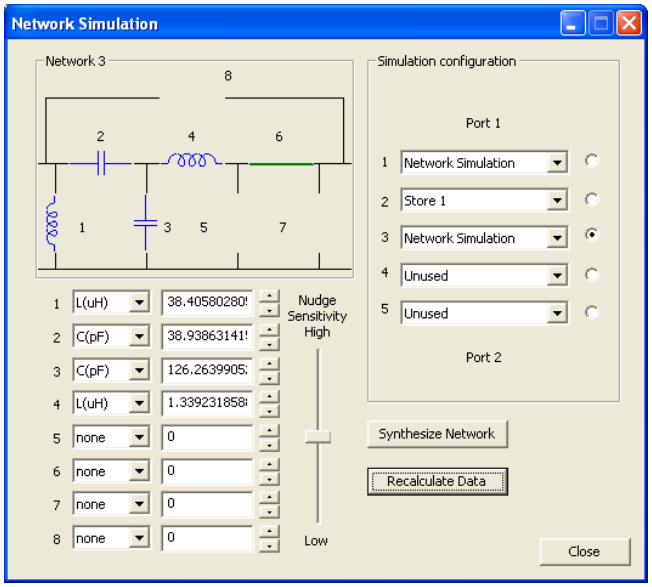
The current list of supported networks is given below:



So for example. selecting an input match of 50 ohms, an output of 900 ohms in parallel with 24pF and a design frequency of 10.7 MHz using a simple L match with series L results in the following display:

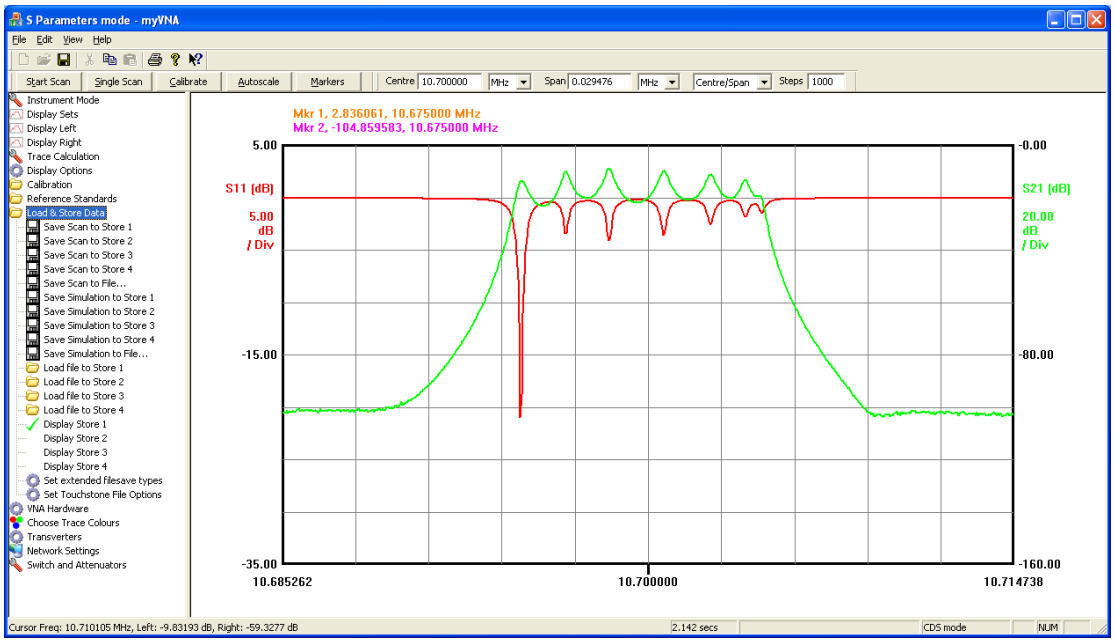


Selecting ‘Store 1” as the second network and generating (just for the hell of it) an LCCL band pass network from 900 ohms in parallel with 24pF to 50 ohms in network 3 results in the following display:

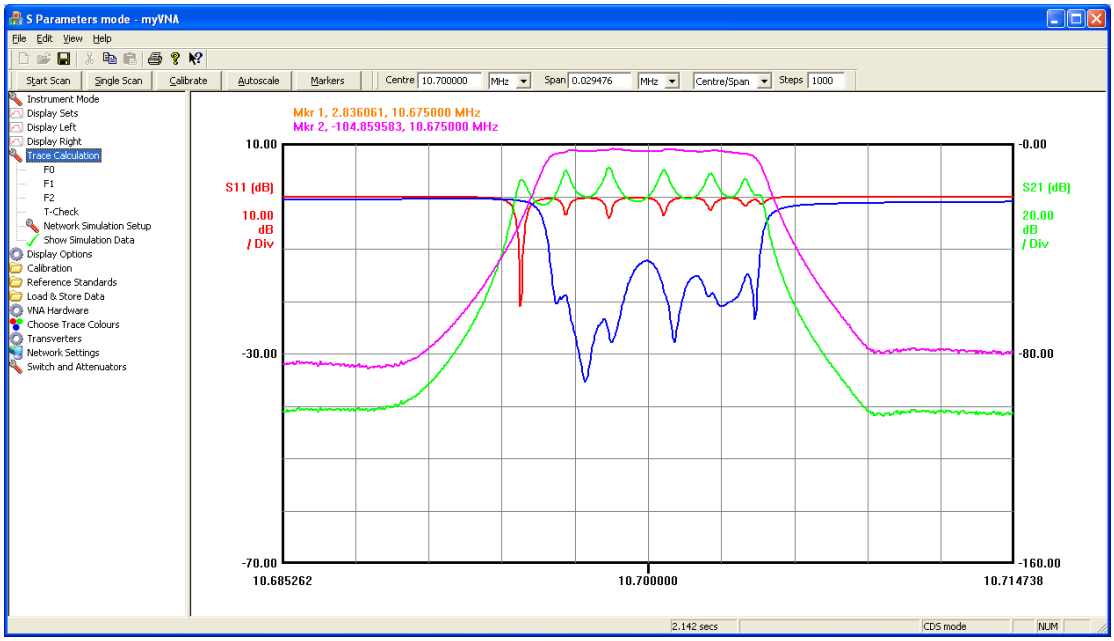


Observe the radio buttons on the right. These permit selection of the network that is shown on the left. As network 3 was the last one edited, that is the one shown. Clicking one of the other buttons causes that network to be shown (and edited) on the left. Also, for convenience, when a network type is changed to "Network Simulation" then that network is automatically selected on the left.

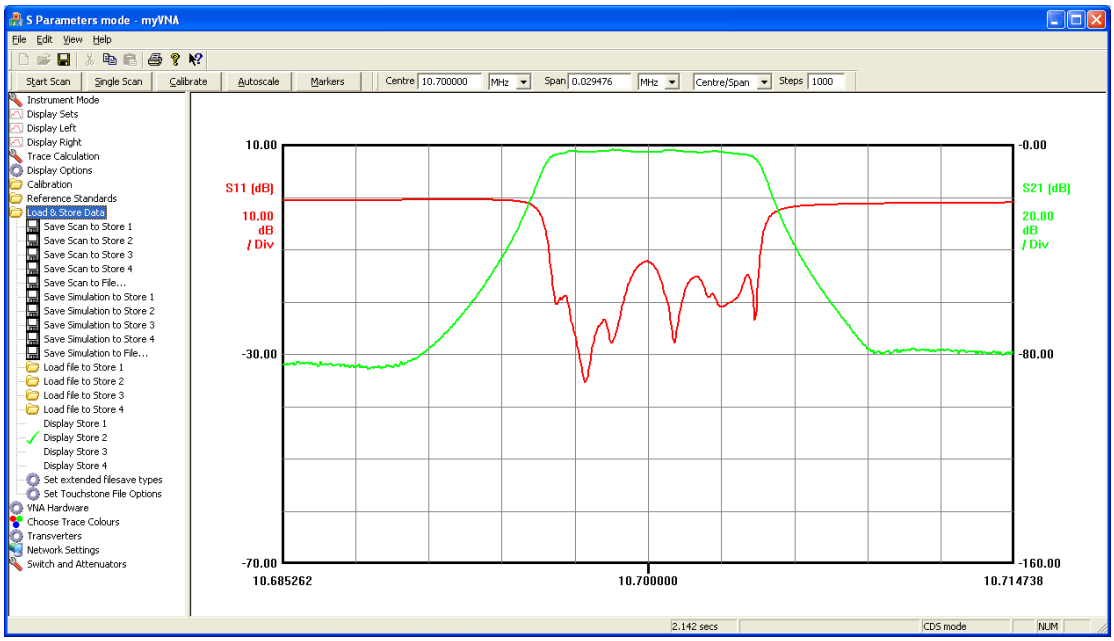
Roderick Wall kindly provided an S2P file that corresponded to a measured crystal filter with a design load impedance of 900 ohms in parallel with 24pF. A step by step example of filter matching is shown in the next section. The trace obtained from it is as shown below:



The trace was obtained by dragging and dropping the s2p file onto the application main window (or it could have been loaded via the "Load/Store Data" controls as shown above). Returning to the "Trace Calculation" option and ticking the "Show Simulation Data" then displays the results of the simulation.



The display above shows the raw store data and the simulation results. The store display may be turned off – it is still used in the simulation, and the display then shows just the simulation:



The resulting simulation data may be viewed as any scan or store data by selecting any desired display type (Reflection parameters such as Rs, Xs, Cs, VSWR etc., transmission or S parameters). The simulation data may also be saved to file in any of the file formats or may be stored in a data store. You may also save it to a store currently being used in the simulation but expect some weird effects.

It is also fun to select a matching network – for example network 1 as defined above – and analyse it. Set networks 1 and 2 to unused and then display the simulation results. What you get is the analysis of network 1. In the above case, selecting a reflection

parameter display with parallel C would show an impedance of about 900 ohms in parallel with 24 pF, varying with frequency of course. It is also possible to display the reflection parameters such as Cp for port 2 by changing the port that is used with the option “Reflection Data Port” under “Instrument Mode”.

On each of the two simulation dialogs as shown above you may have noticed up/down controls next to each value entered, and a slider for “nudge sensitivity”. Clicking on the up or down arrows next to any parameter will nudge the value up or down slightly. How much the value changes by is controlled by the ‘sensitivity’ slider control. When you nudge a value up or down, the synthesis data is recalculated immediately (no need to close the dialog or press “recalculate”. The displayed traces should update for each nudge.

29.1 Load & Save Simulations

The network simulation dialog provides the options to save and load a simulation. Clicking on the Save button opens a file save dialog. This permits a simulation setup to be saved to a text file. Similarly the Load button permits a saved configuration to be loaded.

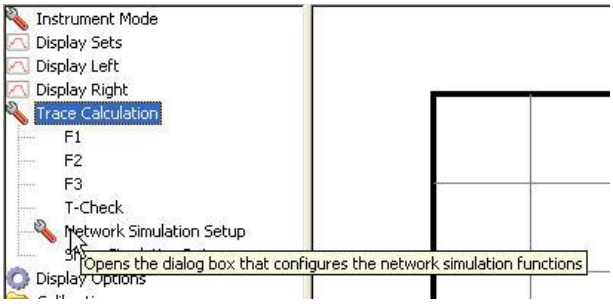
29.2 Impedances

The simulations, by default, use the analyser system reference (normally 50 ohms) set in the “Hardware” section. Sometimes, for example if a filter match is being simulated where the matching impedance is not 50 ohms, it is desirable to be able to specify a different impedance. This may be done using the ‘Network Simulation Port Impedances’ dialog in the network simulation control. This code is subject to change in later versions.

29.3 Filter matching example

The inclusion of the simulation tool was in response to a request from Roderick Wall for filter matching functionality. The following is his description of how to use it (with some tweaks to the text). The previous section used it as an example to illustrate the functionality present; here the same example is given again as a step by step example.

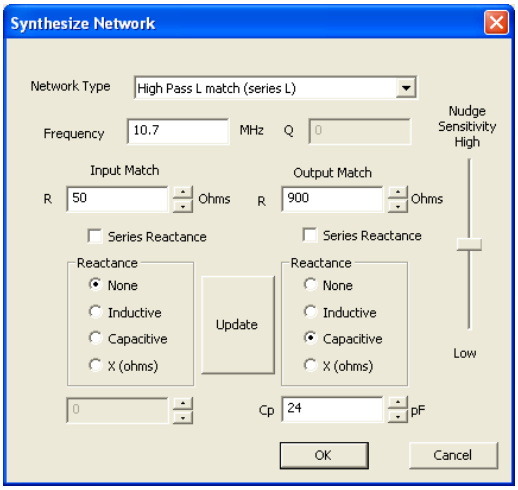
The example used is for an ITT 10.7MHz filter (DUT). The matching requirement is 900ohms with 24pF parallel. The simulation tool can be used to determine the actual filter, to check filter performance and to determine the component values for the matching circuits. The ITT Filter s2p data can be either scanned data or stored data. For this example the s2p S-parameter data is in “Store 1”; but the data could equally be from live scan data.



Under "Trace Calculation" click (select) "Network Simulation Setup". We will use the simulation tool to generate two matching networks. A network from 50 ohms to 900 ohms with 24pF parallel capacitance and from the same 900 ohms with 24pF parallel back to 50 ohms.

The simulation tool will be configured to use 3 of the 5 available networks for this function, with networks 1 and 3 used for the matching networks and network 2 used for the measured performance of the filter by the VNA in a 50 ohm system. There is nothing special about the choice of networks; we could equally have used networks 1, 4 and 5. The important point to note is that the sequence is as shown; the networks are in series from port 1 (network 1) to port 2 (network 5).

Select "Network Simulation" for the input matching circuit for port 1 of the Filter (DUT). Note, the matching circuit component values will only be shown after the next step (Synthesize Network). Click the "Synthesize Network" button to show the following window.

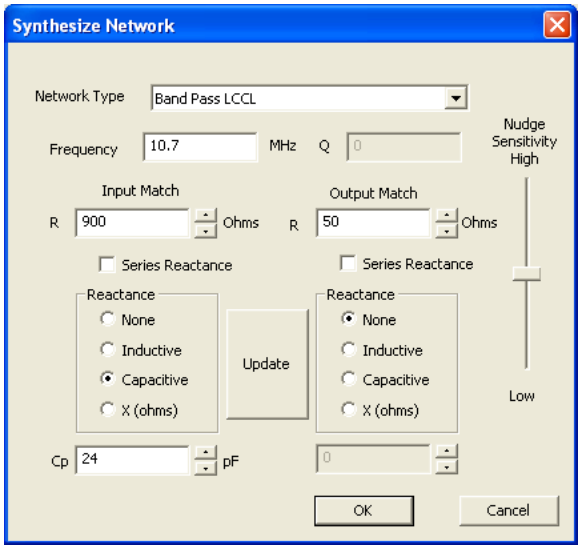


Select the "Network type" and enter the matching circuit input/output impedances. Click "OK". You are free to choose whichever type of matching network you prefer. By way of an example, the first network used here is a simple high pass L match.

Select the location of the ITT Filter (DUT) s2p S-parameter data, it can be either scanned or stored data. For this example the ITT Filter s2p S-parameters are in "Store 1". If you don't have an s2p S-parameter filter file, load the example "10.7MHz Sample Filter.s2p" file into "Store 1". This file may be found in the installation directory of myVNA. Depending on the program configuration when you load the

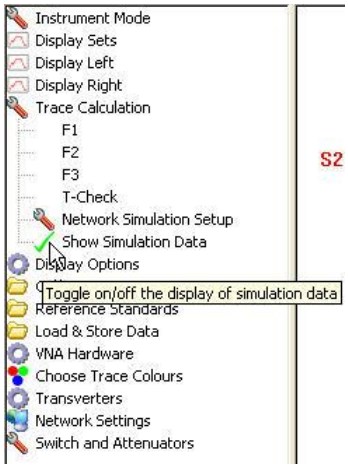
file, the scan parameters may be automatically set to match that of the data loaded; if not set the centre frequency to 10.7 MHz and the span to 30 kHz.

Select "Network Simulation" for the output matching circuit for port 2 of the Filter (DUT). Note, the matching circuit component values will only be shown after the next step (Synthesize Network). Click the "Synthesize Network" button to show the following window.

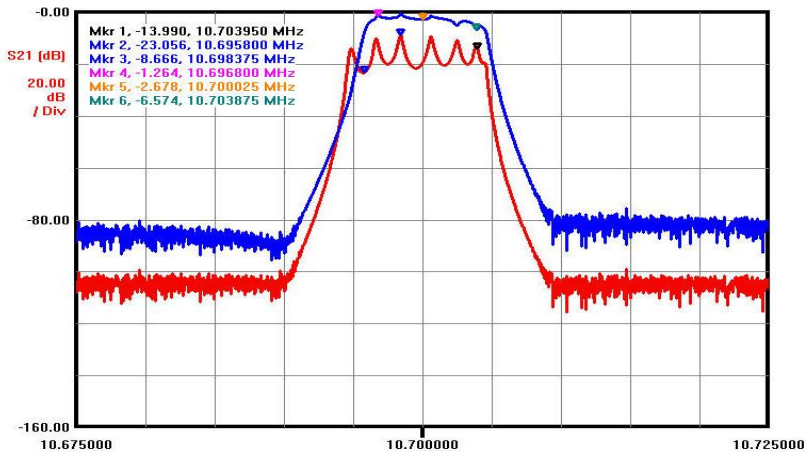


Select the "Network type" and the matching circuit input/output impedance. Click "OK".

From the Network Simulation window click "Recalculate Data". This will generate a simulation of the above 3 networks. To display the simulation trace select "Show Simulation Data" option under "Trace Calculation" as shown below.



The simulation trace will be displayed with whatever display modes are selected. For example, selecting "S21" should yield the following display:

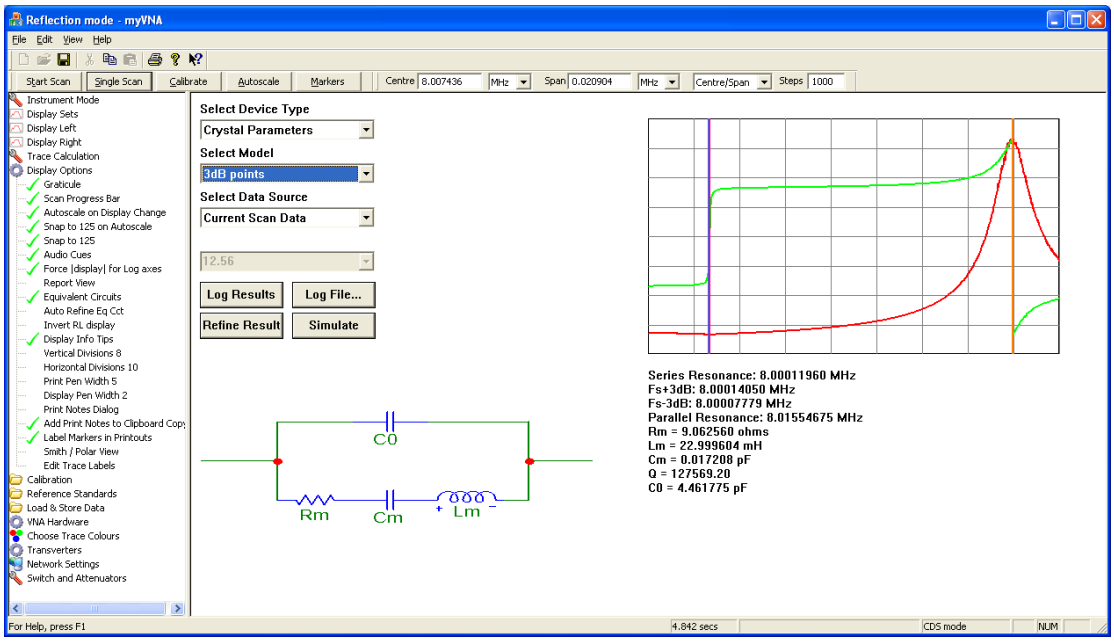


S21 Traces, unmatched (red), matched (blue).

29.4 Simulation and Crystal Equivalent Circuits

Simulation support has been added to Crystal Equivalent Circuits. Don't try to use it with the 6 term model – it is not going to work.

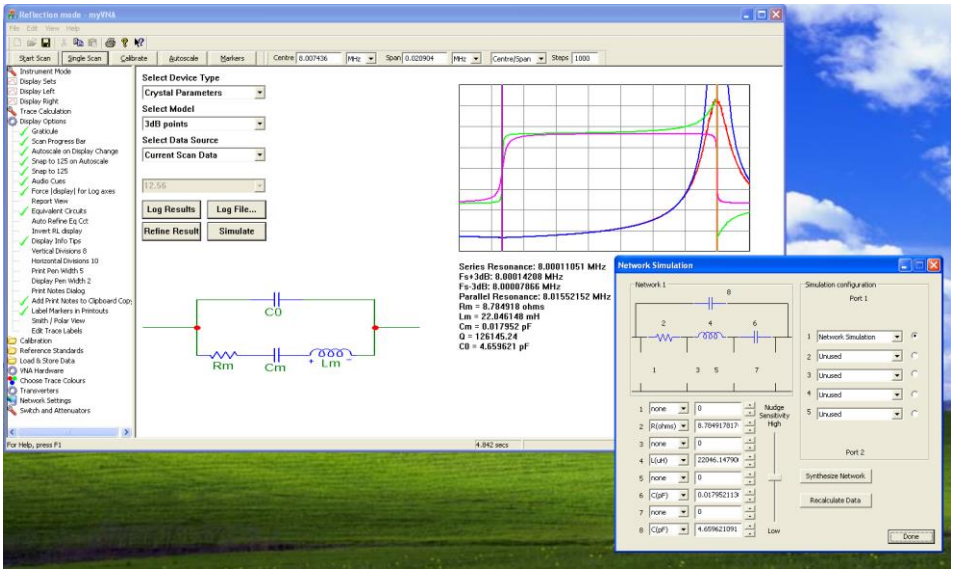
Under crystal simulation, there is a new button "Simulate"



If this button is pressed, the current simulation configuration is replaced by that of the crystal parameters being shown.

The simulation control dialog does not have to be open for this to work, but if it is (or if it is opened later) then the schematic of the crystal will be shown

The results of the simulation will also be displayed on the screen with the main trace as shown below. In this example, the green and red traces are the scanned data, the other two the simulated data from the results.



29.5 Simulation Limitations

There a couple of limitations to be aware of. First if scan and store are used at the same time the number of data points in the scan and store must be the same and the frequency details must be the same. The number of data points in the simulation will adjust itself to the scan or store settings automatically.

Now the second limitation. Simulation uses ABCD (transfer) scattering parameters, and to work correctly this means that the program must possess forward and reverse dual detector data. The program will try to be helpful, but be careful. If you do not have valid reverse data for a network, it will reuse the forward data for the reverse direction in order to have something to use, but this assumes the network is symmetrical.

30 Multiple Program Configurations

Multiple program configurations are supported. Specifically, the registry entry that the program uses is the same as the executable (normally myVNA.exe hence the registry entry would be myVNA). If you make a copy of the program, for example myMSA.exe, then a separate configuration will be created – each will have its own settings.

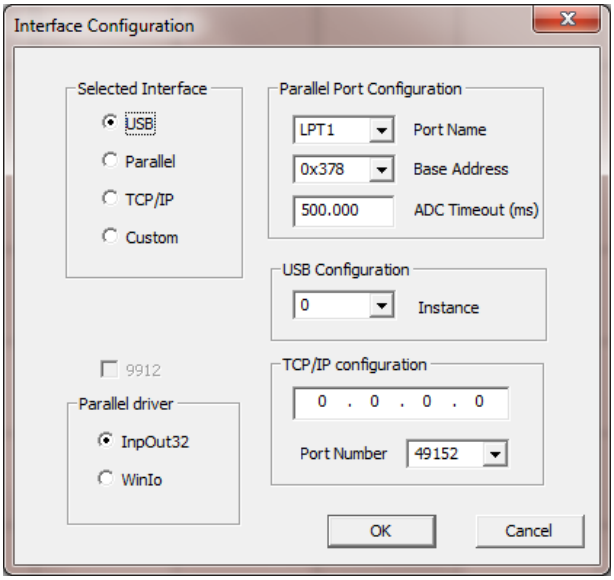
This may be used to create multiple configurations for one piece of hardware or may be used to create different instances for different hardware types such as for the N2PK and the MSA.

An alternative to multiple configurations is to export & import the configuration using the options under “Instrument Mode”

31 Multiple USB connected VNAs

Multiple USB connected VNAs may be controlled at the same time from one PC. In general this is not in general recommended unless the scan is slow or the PC is fast as it will probably run slowly.

Under “Hardware Interface” is an option to select the USB instance:



Each time a USB device is recognized (which is usually in the sequence they are connected but don’t count on it), it is allocated an instance ID.

Instance IDs are not saved & reloaded when the program starts – it makes no sense to do so.

Start an instance of myVNA for each hardware VNA but note that they will share the same configuration – if you do not want this to happen, make copies of myVNA.exe and give each executable a different name.

For each instance, select an instance from the “USB Configuration” control as shown above.

32 OLE Automation

You can control myVNA and through it the VNA via Microsoft Object Linking and Execution (OLE). This chapter shows how to do it.

The program exports an automation interface for remote control. The intention of this is to permit an external program to control program operation. Not all functions are available; the intention is to permit control of the GUI and export data not to try to expose all methods; there is little point for example in providing external control of the ability to drag a cursor on the screen. However if I missed something you need by all means drop me an email.

The interface is described by the file myVNA.tbl which is to be found in the installation directory of the program. As an introduction, in Appendix D is the .odl input file of the (current) version of the interface that produced the .tbl file. The details of the parameters needed are shown in the AccessMyVNADll.h file (see the next paragraph) and are also shown in Appendix D.

The best way to use the automation interface is to access the OLE interface directly as defined in the tbl file, however as an alternative, a companion DLL has been produced that provides a basic interface. The use of this is illustrated in a companion project "AccessMyVNA" that comprises the DLL and a dialog box project that is linked to it. To show the use of the automation interface, run AccessMyVNA.exe. This should display the dialog box. Now click the "Init" button. This should cause an instance of myVNA to be automatically executed (and minimised) and some of the values to be automatically loaded. It may then be controlled by means of the other controls on the dialog.

So, how does the interface get registered? During installation the installer runs myVNA with command line switches of "/Register" and "/Unregister". Executing these will register or remove the Automation interface from the windows registry. Unfortunately unregistering will also clean out some of the registry information that the program uses; the only implication as far as I can see is that when next it is run the toolbar positions will have reset themselves.

33 Network Client / Server

The network client / server code is a precursor to a possible Ethernet interface to the VNA and permits remote operation of a VNA on another PC. This chapter describes its use.

The idea is to allow control of the VNA via an Ethernet interface. This interface would either be an adjunct to the USB interface or (for now) another PC running myVNA. In other words for now it just provides remote control of the VNA connected via USB to one PC from another PC – which could either be local or anywhere via the Internet.

Some care is needed – the code has limited error handling.

To use it, the VNA is connected by USB – it must be USB – to one PC. That PC runs myVNA and is configured to act as a server. Another PC also running myVNA has its hardware interface set not to Parallel or to USB but to an IP address. Once that is done, functions such as "scan" on the second (client) PC will cause the server PC to perform a scan and the result will appear on the screen of the client. Note that the calibration data and configuration used are those of the client PC not the server – the server is just providing remote access to the USB connected VNA.

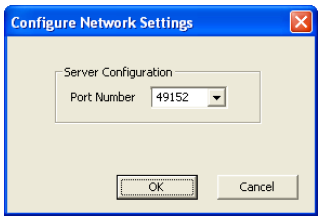
To set a PC up as a server do the following

1. Configure the PC to use USB interface and make sure that it is working correctly
2. Configure the server details as shown below
3. Switch on the server mode

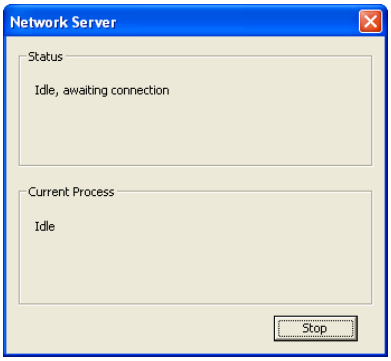
To set up a client PC do the following

1. Set the interface type to TCP/IP with the IP address of the server
2. Perform a scan, signal generator or other function

To configure a server, open the "network settings" on the left hand treeview and select "configure remote access". The following dialog will appear

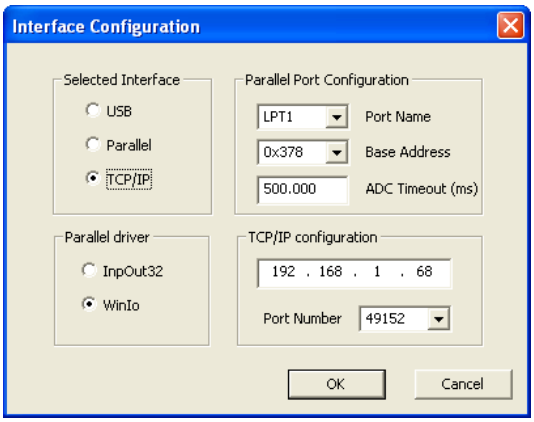


Set the port number to the desired IP port (or just leave it at 49152 if you don't understand this)
To start the server running, click on the "Accept Remote Connections" option. The following dialog will appear



The Status message will change when a remote client connects, otherwise it indicates Idle. The Current Process will not show any information in this version of the code

To configure a client open up the “Hardware Interface” under “VNA Hardware” and set the IP address to that of the client, set the port number to match and select the TCP/IP option



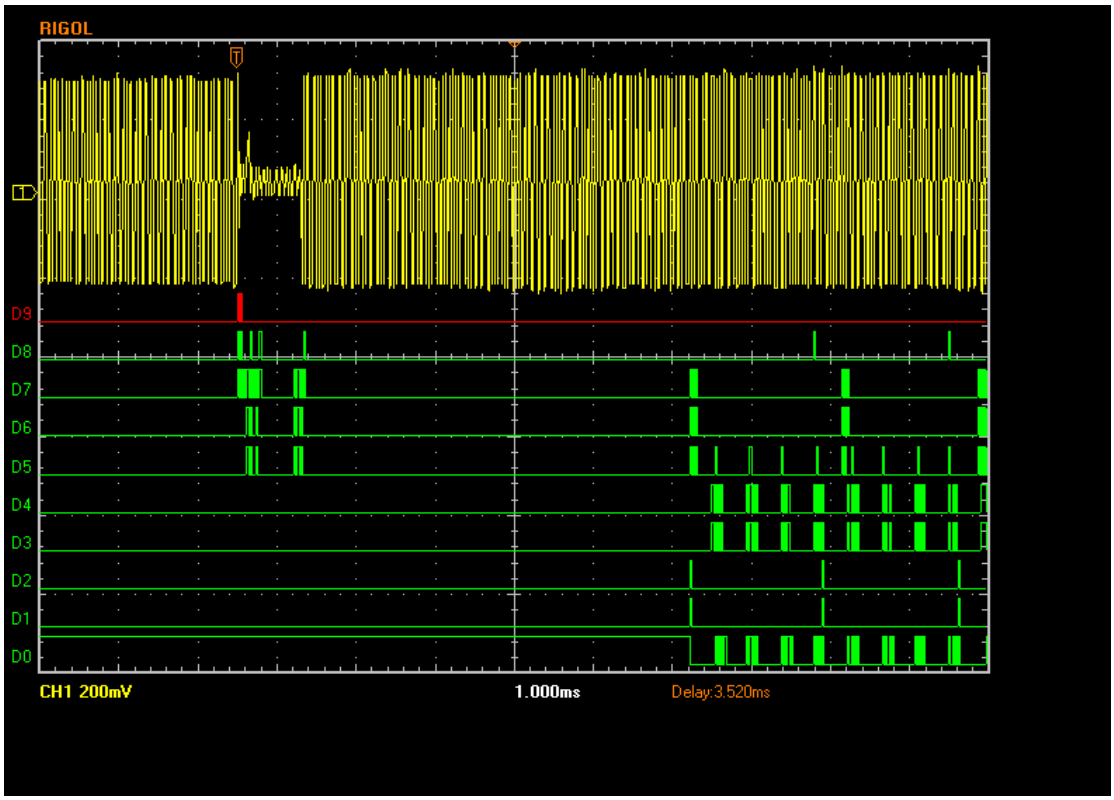
That’s all there is to it. There are a number of new error dialogs etc.

Only IP V4 is supported and firewalls may stop connectivity.

34 DDS Power Down

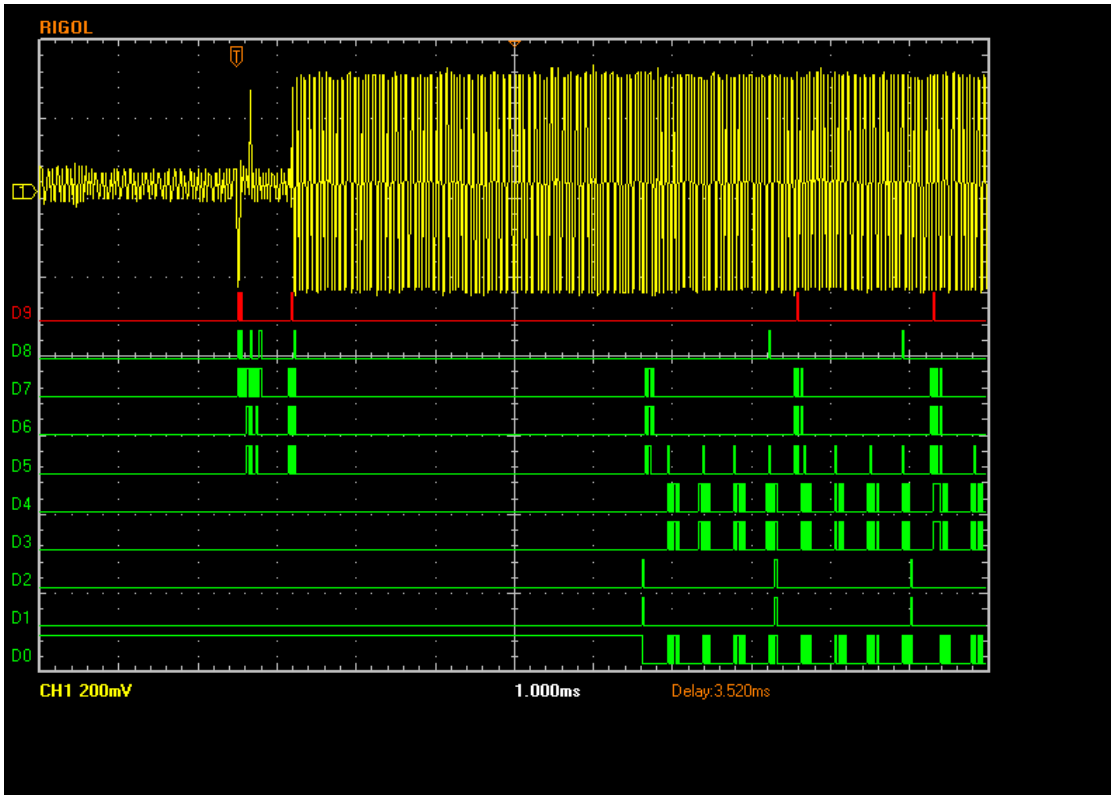
The DDS chips may be powered down when not in use or left powered up. Under "VNA Hardware" is an option that controls this. When selected, the DDS chips will be automatically powered down when not in use reducing current consumption from 0.25 to 0.3A to 0.15A at 12V.

Here is a trace without power down. The trigger point is the start of the sweep. DDS reset occurs, switching off the RF outputs for about 0.5 ms before the signal reappears (the yellow upper trace). In this example, there is a 5000 µs sweep start delay which can be seen before the first point is read, and the RF outputs are held steady for this time.



When DDS power down is selected, the DDS chips are put into power down mode at the end of a sweep and held there until needed. This may be seen in the following trace.

This power down occurs after all activity; the end of each sweep, between calibration runs, when exiting VVM or Signal Generator modes etc.

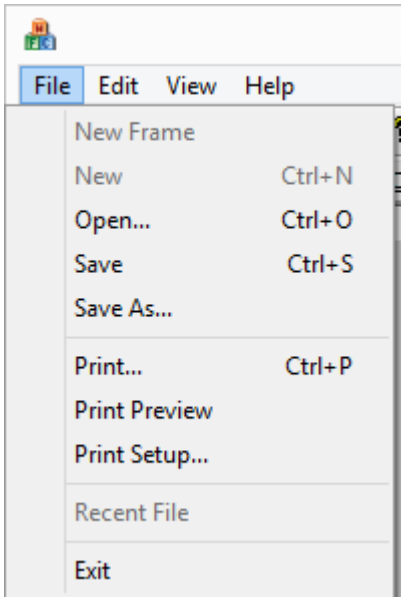

RIGOL
D9
D8
D7
D6
D5
D4
D3
D2
D1
D0
CH1 200mV
1.000ms
Delay:3.520ms

35 Command Reference

This chapter provides a brief summary of all commands

35.1 Main Menu

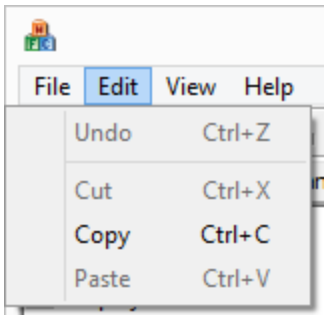
35.1.1 Main Menu - File



The main menu file commands provide the following functions:

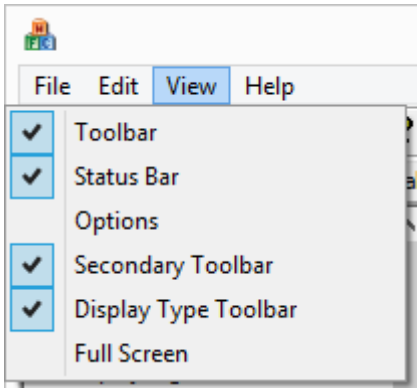
- Open. Used to load a saved trace from a file (same as the file open icon from the windows toolbar or the keyboard shortcut control-o). It will prompt for which data store to load the file into and will then prompt for the file. Supported file types are .csv, .s1p and .s2p. The program will attempt to interpret the contents of the file. For more information see Section 6.1 “Saving Data to and loading data from Disk”.
- Print. The current data that is displayed is sent to the printer via a standard print dialog (same as the print icon on the windows toolbar or the keyboard shortcut control-p). For more information refer to Section 6.2 “Printing”.
- Print Preview. The current data that is displayed is formatted and displayed as it would be printed given current printer settings. For more information refer to Section 6.2 “Printing”.
- Print Setup. Opens a standard printer setup dialog.
- Exit. As it says, exit the program
- All other functions on this menu currently have no function.

35.1.2 Main Menu – Edit



The only function available from this menu is “Copy” which copies the current display to the clipboard (same as the copy icon from the windows toolbar or the keyboard shortcut control-c). For more information see Section 6.3”Copy to Clipboard”.

35.1.3 Main menu – View



The View menu has three separate functions:

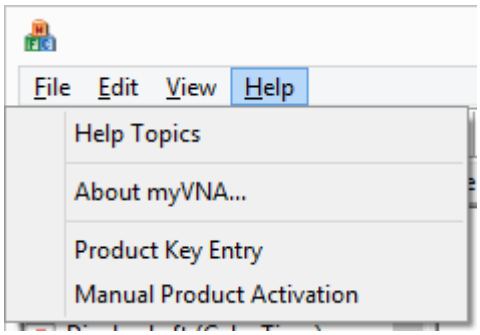
- Determine which toolbars are displayed
- Set the program options
- Switch to full screen display mode

The toolbar, status bar, secondary toolbar and display type toolbar may be switched on or off. The scan control and span toolbars and the main menu bar may not be hidden.

Options set certain limits and configuration options that affect the program behaviour. Details are given in Section 7.

Full screen display is self-evident. To exit full screen mode, hit the “Esc” key.

35.1.4 Main menu – Help



The Help menu provides access to the help file (pressing F1 does the same thing as do the two help icons from the windows toolbar). “About myVNA” gives program authorship and copyright information. The final two entries are used to manage product keys and authorisation.

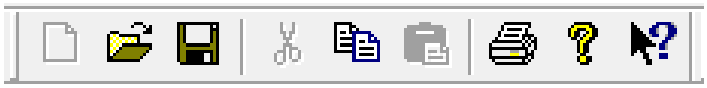
A product key is needed in order to run the program, and the dialog box used to enter, view or change it is accessed from here. Depending on the product key, product activation is usually required in either 30 or 60 days from when the product key is entered. Full details are given in Sections 3.2 and 3.3.

35.2 Toolbars

There are six toolbars supported by the program. One of these is only present in MSA mode.

All toolbars are moveable and dockable. Their location is preserved when the program is closed.

35.2.1 Windows Toolbar



The windows toolbar provides quick access to the following functions:

- Open a file (same as the menu command “File / Open…” or the keyboard shortcut control-o). It will prompt for a data store to load a trace into then open a file open dialog to permit a csv or snp file to be loaded.
- Copy to clipboard (same as the menu command Edit / Copy to clipboard or the keyboard shortcut control-c) – the current display trace together with associated data is copied to the clipboard
- Print the current image (same as the menu command “File / Print” or the keyboard shortcut control-p)
- Open the help file (same as the menu command “Help / Help Topics” or the keyboard shortcut function key F1).

35.2.2 Controls Toolbar

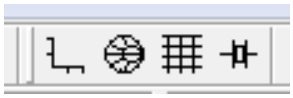
From left to right (or top to bottom if the toolbar is docked on the side of the display) the toolbar provides quick access to the following functions. Unless noted otherwise the same functions are also available from the popup menu.

- Set vertical axis – opens a dialog box that permits the left and right display axes to be configured if in rectangular display mode, or sets the radial axis if in polar display mode. The same functions are also available from the display popup menu under the names “Set vertical axis” and “Set Radial Axis” respectively.
- Set frequency axis – opens a dialog box that permits the frequency axis to be configured if in rectangular display mode, or opens a dialog box to set the frequencies and time settings if in polar display mode.
- >S Set scan to current display. This sets the scan start and stop frequencies to match those of the current display. The display may, for example, have been dragged to zoom in on a feature, this button sets the next scan to match the desired display.
- >DSet the display to match the current scan. This sets the display to match the current scan start and stop frequencies. The display may, for example, have been dragged to zoom in on a feature, this button sets the display back to match the current scan.
- DI “Display Lock” - When toggled on, locks the current frequency axis to the scan settings so that dragging the display left or right will not change the settings, but any change to the scan start / stop will also update the frequency axis.
- LI “Left Lock” - When toggled on, locks the current left axis so that dragging the left side of the display up or down will not change the settings.
- RI “Right Lock” - When toggled on, locks the current right axis so that dragging the right side of the display up or down will not change the settings.
- FI “Frequency Lock” –
- M Bring manual markers onscreen and position them at the centre frequency of the scan.
- Hide / Unhide the markers. If there are any displayable markers on the current trace, this control toggles their display on and off.
- Toggles on or off a logarithmic frequency display.
- Toggles on or off a logarithmic vertical display.
- When pressed, this button clears any reverse scan data.
- When pressed, this button clears any forward scan data.
- EL Edit Trace Labels. The same function is also available from the display options treeview. When selected, it opens up the dialog box that permits the editing of labels associated with the traces.
- Pn Edit Print Notes. The same function is also available from the print options treeview. When selected, this button opens a dialog box that permits the notes to be included on a printout to be edited.
- RI Reset Axis Label Positions. When pressed it moves all axis labels back to their default positions.
- Open and display the marker measurement dialog box. This dialog box allows configuration of what parameters are to be displayed or printed against each marker.
- tdr Opens the dialog box that permits the Time Domain Reflection (TDR) mode to be configured.

- When selected, ths control opens the trace averaging dialog box.

35.2.3 Display Format Toolbar

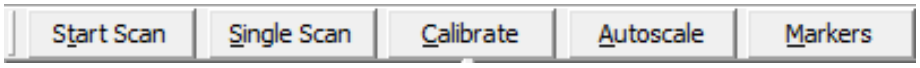
This toolbar permits selection of the main display modes, rectangular, polar, report and equivalent circuit.



The same functions are also available from the display options treeview controls

35.2.4 Scan Control Toolbar

The scan control toolbar is the primary control for starting and stopping a scan as well as autoscaling the display and configuring markers.



The first button starts or stops scanning. The second performs a single scan. When scanning repetitively, a scan may be completed then stopped by pressing the single scan button.

The third button starts calibration by opening a calibration dialog.

Autoscale will cause the display left and right axes to rescale themselves to match the current display configuration and trace data.

The final button will open the marker configuration dialog box.

35.2.5 Span Toolbar

The Span toolbar controls the start and stop frequencies for a scan, the number of frequency steps and the number of measurements to be taken and averaged at each point.

The dropdown menu shown as “Start/Stop” permits the mode of data entry to be changed. The labels and functions of the preceding two controls (labelled here Start and Stop change accordingly. The units for a frequency may also be changed for each to be Hz, kHz or MHz.

The modes of entry supported are:

- Start and Stop frequency
- Centre frequency ad span
- Full scan (analyser minimum to analyser maximum)
- From Zero (same as full scan but the display starts from zero to avoid unusual settings per division on the display

- Centre / per step. The second frequency box determines the frequency change for each step between points
- Centre / per division. The second frequency box determines the frequency change per division on the display
- Start / per step – same as Centre / per step but the first box is the start not the centre frequency
- Start per division – same as Centre / per division but the first box is the start not the centre frequency

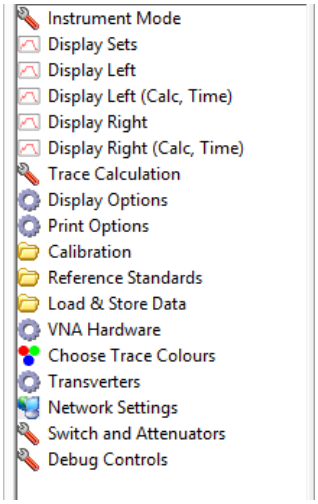
35.2.6 MSA Path toolbar

The final toolbar is only present in MSA hardware mode. It controls the "Path", or filter settings, frequency offsets for the filter and the video bandwidth setting.

35.3 Treeview controls

The treeview on the left provides quick access to detailed settings. Many of the more common controls are also available on the controls toolbar or popup menus

The treeview contains the following main settings



Whenever a treeview node is opened, any previously open node is closed in order to keep the display manageable.

Not all of the controls may be visible. The Debug controls are normally hidden; they may be displayed by unticking the "hide debug controls" options setting. In addition options dialog permits the following to be hidden if you do not need them:

- Display Sets
- Transverters
- Network Options
- Switch and Attenuator controls

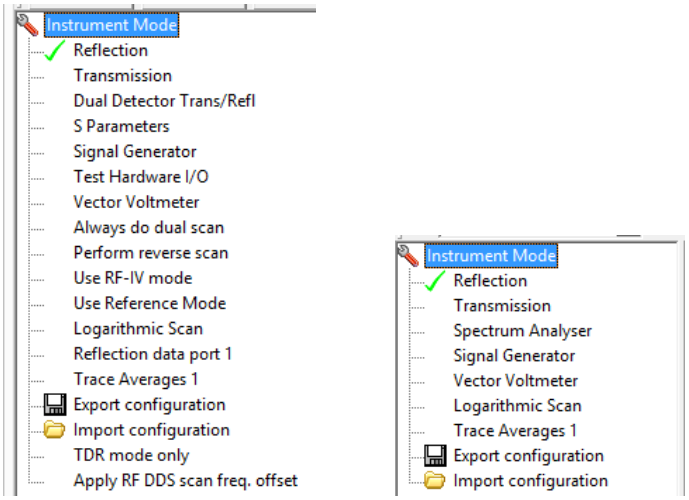
Finally, the options shown under left, right and display sets normally changes dynamically to reflect the scan mode – there is little point usually in displaying transmission gain options if a reflection scan is performed, however there are

occasions when it is very useful. There is also the option to display all types regardless of the mode.

35.3.1 Instrument Mode Treeview Node

The Instrument Mode treeview node controls the main characteristics of a scan.

The Instrument mode treeview control takes one of two forms depending on whether the hardware type is set to MSA or N2PK. The MSA set is a reduced set but includes spectrum analyser as a mode. Below are shown both options, the N2PK on the left; the descriptions refer to the N2PK case.



The “reflection”, “transmission”, “Dual Detector trans/refl” and “S parameters” work as a set, only one may be ticked at a time. The setting controls what type of a scan is performed as well as the display options that are presented. Unless “Always do dual scan” is ticked, then when a reflection or transmission scan is performed, only one ADC is used and the scan data will contain only transmission or reflection data. In all other cases, the scan includes data from both ADCs, and in this case once data is available, you may switch freely between the modes to change the display without updating the scan data. If the calibration mode is unsuitable for the selected mode, the program will issue a dialog box informing you of the mismatch and will not perform a scan.

The two controls “Vector Voltmeter” and “Signal Generator” open dialog boxes for the vector voltmeter and signal generator.

“Test Hardware IO” is a diagnostic mode used to check for hardware connectivity faults. For more information, refer to Annex G -

“Perform reverse scan” is used to instruct the VNA that, when reverse scans in S parameter mode are being performed manually, that subsequent scans are reverse mode.

There are two specific modes of operation that may be selected in addition to the main scan modes, and these modes interact with the scan mode as not all permutations are

supported. RF-IV mode and Reference Mode use an external hardware module to measure V and I independently and improve measurement accuracy. There is also specific hardware configuration that needs to be set to define the way that the hardware module is used and controlled, for more information see sections 21 and 22.

“Logarithmic Scan” changes the scan mode so the scan points are distributed between start and stop on a logarithmic basis rather than a linear one. This scan setting is separate to the display mode (log or linear) but when changing the scan mode, the display mode is also updated to reflect the scan setting. The display mode may subsequently be changed as desired independent of the scan mode.

“Reflection data port” is a shortcut override useful in some circumstances where the port used for reflection mode measurements is swapped. Be careful though as it does not change the transmission setting, and can therefore cause a scan not to be made if both ADCs are needed.

“Trace Averages” performs the same function as the trace averages control on the span toolbar – it sets the number of readings taken at each scan step which are then averaged.

The next two entries – export and import configuration – permit the current program settings to be saved to file or reimported, useful if specific configurations are frequently used. The file that is saved and loaded is a text file with human readable values but note that the program expects specific defined values and formats – do not expect it to interpret any setting you may care to make in another format to the one expected.

“TDR Mode Only” is used when using TDR mode as a means to suppress the display of frequency domain traces leaving just time domain information.

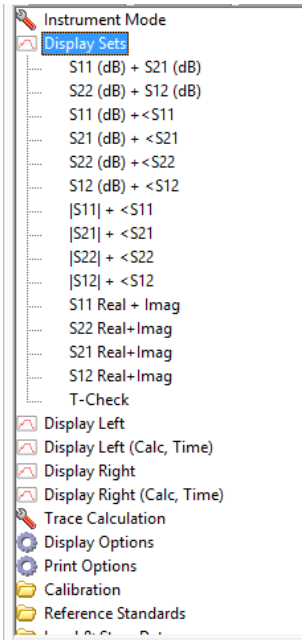
Finally, “Apply RF DDS scan freq. offset” is an experimental mode that adds a fixed frequency offset to the RD DDS frequency setting. The frequency offset is defined in the hardware settings.

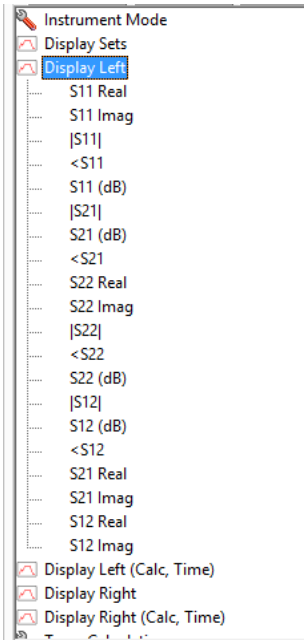
35.3.2 Display Treeview Nodes

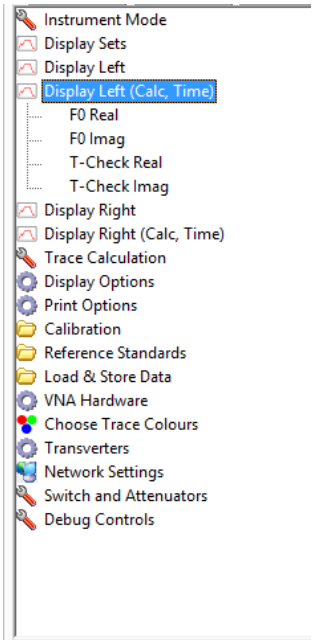
There are five treeview nodes that change the parameters displayed:

- Display Sets
- Display Left
- Display Left (Calc, Time)
- Display Right
- Display Right (Calc, Time)

All work in the same fashion. When opened, each shows a list of parameters – for example as shown here (three are shown side by side, the left and right options are identical)

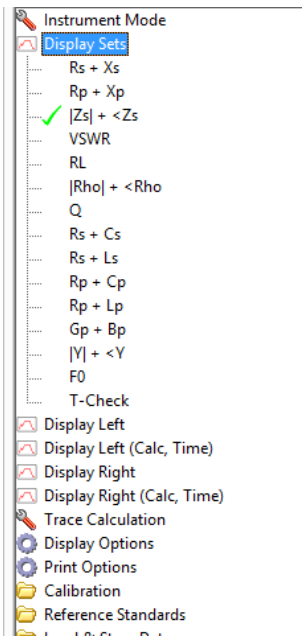


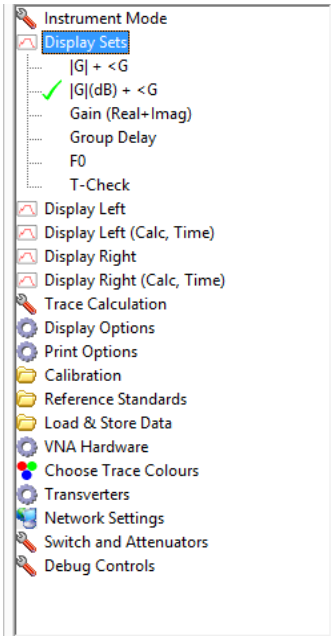


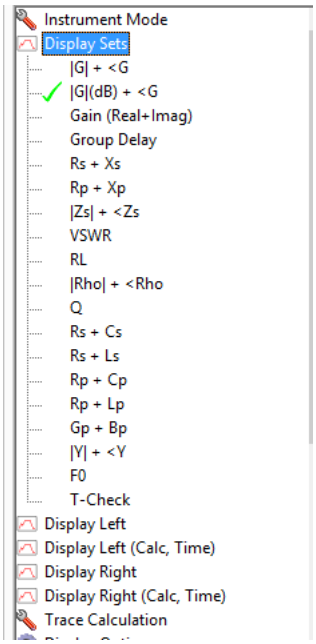


Parameters are selected by clicking them. To select more than one at once, hold down the control key when clicking on a setting. The parameters selected will be shown for all selected race data in the display. Select many and it can get hard to read though. “Display Sets” represents a set of shortcuts for setting left and right axis displays at the same time – for example selecting “S11 (dB) + S21 (dB)” achieves the same as selecting “S11 (dB)” on the left display and “S21 (dB)” on the right display.

The choices offered in the display nodes is conditioned by the instrument mode chosen – in this case it was set to S parameters. Selecting Reflection, Transmission or dual detector yields different choices – for example as shown below:





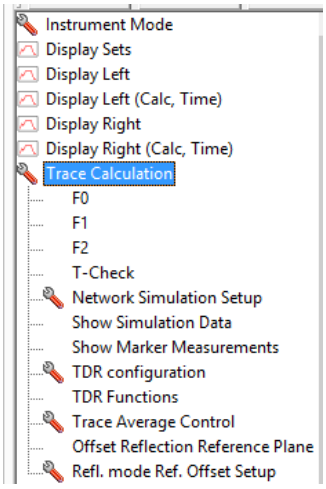


It is also possible to display all choices at all times regardless of the mode by setting the “Show all display types at once” option in the Options dialog.

35.3.3 Trace Calculation Treeview Node

The trace calculation treeview node provides access to:

- the user defined functions,
- network simulation,
- marker measurements
- Time Domain Reflection (TDR) controls,
- trace averaging and
- reflection mode reference plane offsets



The first four entries (F0 to T-Check) open up dialog boxes to configure the four user defined functions. The default names for each are F0, F1, F2 and F3, but the dialog box permits the name of the function to be set – the final entry in the example above has been set to "T-Check". When a valid equation is set for one of these functions, it will then appear in the lists of display sets, left or right options and may be used to display trace data. More information is given in Section 28.

The next two entries control network simulation. The first opens up the dialog box to configure network simulation, the second toggles on or off the display of the results. Refer to Section 29 for more information.

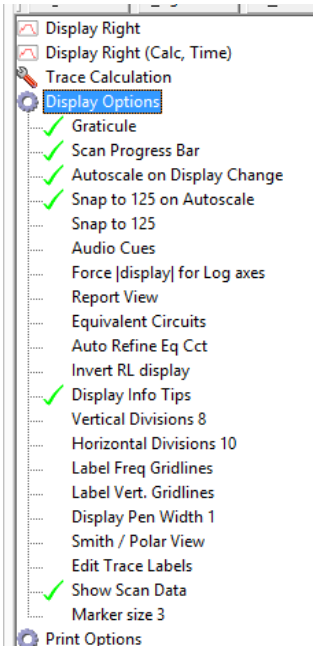
"Show Marker Measurements" displays the marker measurements dialog.

The next two nodes control TDR operation. The first opens the TDR configuration setup dialog, the second toggles on or off the TDR results display. TDR mode is addressed in Section 23.

"Trace Average Control" opens the dialog box that controls trace averaging. See Section 25 for more details.

The final two entries manage the application of an offset to the reflection mode reference plane; the first toggles on and off the application of the offset, the second configures the offset. This feature is experimental – if you find problems please let me know.

35.3.4 Display Options Treeview Node



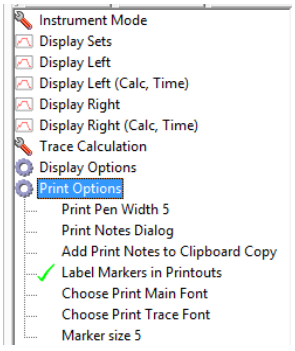
This node controls many of the functions of the trace display pane. Many of the options are self explanatory and simply toggle on or off the appropriate setting.

The current display mode is set by selecting one of "Report View", Equivalent Circuits" or "Smith / Polar View". When none of these are selected, rectangular display mode is shown.

The "Auto Refine Eq Cct" setting will cause the refine function to be executed every time a scan is performed in equivalent circuit mode.

The "Invert RL display" option inverts return loss display – personal preference if you wish.

35.3.5 Print Options Treeview Node

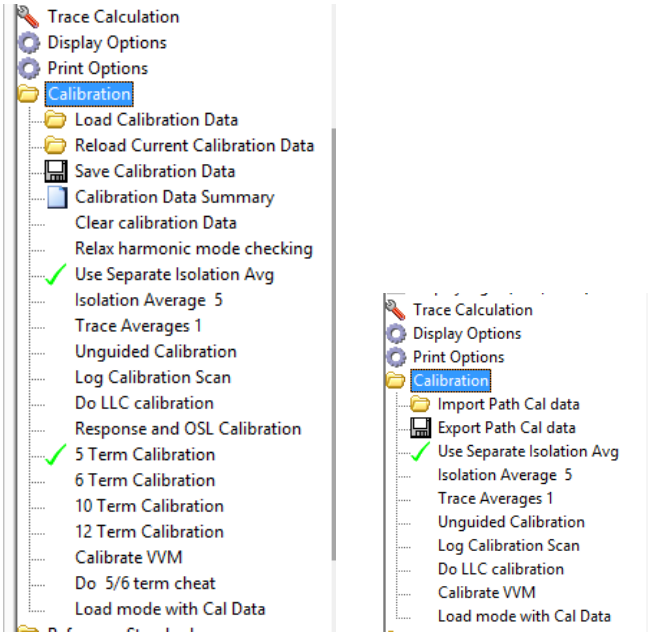


The print options are display options that apply only to the print or clipboard copy functions. Again, all are self explanatory. Print notes provides the facility to add header, footer and notes to a printout, and this may optionally be include in the clipboard copy. Printing is addressed in Section 6.2;

There is also the facility to change the font used as well as the size of the markers and the pen width.

35.3.6 Calibration Treeview Node

This node controls all the main settings associated with calibration. On the left is show the N2PK case, the MSA case is a reduced set and is shown on the right. The descriptions refer to the N2PK.



The first three settings are associated with loading and saving calibration data – Reload reloads the last saved or loaded file.

Calibration Data Summary displays a summary screen of the currently held calibration data.

Calibration data may be cleared using the “Clear” function.

When using harmonic modes, the program will normally not permit mixing of modes and calibration data as this will result in strange results, but for test purposes it is sometimes useful. The “Relax harmonic mode checking” function permits this to be overridden.

When performing calibration, the isolation reading in transmission mode is the one most likely to suffer noise, the next two controls permit a separate average to be made of the isolation calibration scan readings.

Separate from the main trace averaging, calibration scans may also be averaged using the “Trace averages” control.

There are two main forms of calibration – guided, where the user is taken step by step through the calibration process, and unguided where calibration is performed as required. The mode is set by ticked or unticking the “guided calibration” control.

A calibration scan can be either logarithmic or linear in its steps. It does not really matter whether it matches the scan mode setting as it will interpolate but it is preferable if they are the same, especially if the number of points is small.

Low Loss capacitor (LLC) calibration is an alternative to Open Short Load (OSL) calibration and is performed if this option is ticked.

The next five entries ("Response and OSL Calibration", 5, 6, 10 and 12 term) define the calibration mode – only one may be ticked at a time. For more information please refer to Section 13.

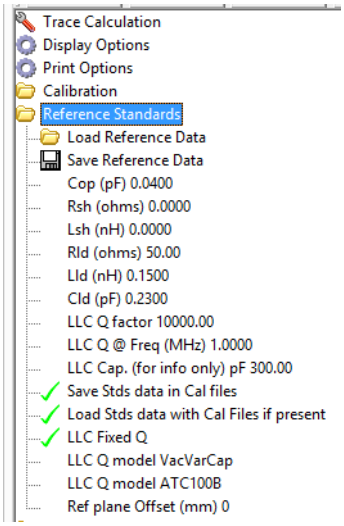
Calibrate VVM opens the calibration dialog box for the VVM mode, which is addressed in Section 27

Selecting the "5/6 term cheat" applies 10/12 term calibration but it just uses the 5/6 term calibration data.

Finally, selecting the "Load mode with cal data" will cause the mode to be set automatically with whatever calibration data is loaded.

35.3.7 Reference Standards Treeview Node

This node provides access to the calibration data settings. For information on reference standards please refer to Section 18.



Reference Standards data is a set of parameters used during calibration to correct for specific characteristics of reference devices (Open, Short, Load, LLC). For further information see Section 18.

The first two entries in the list permit the complete set of reference standards data to be saved or loaded. The file is a text file so can – with care – be edited.

The next 6 entries define the errors in the Open, Short and Load. For the open reference, there is just one parameter – the capacitance incurrent by fringing. For the

Short reference, there are two parameters, the series resistance and inductance, as fringing capacitance will be negligible. For the load reference, the series resistance and inductance together with the parallel capacitance must be given. The default values are those that may be expected of a simple SMA female connector constructed set of references.

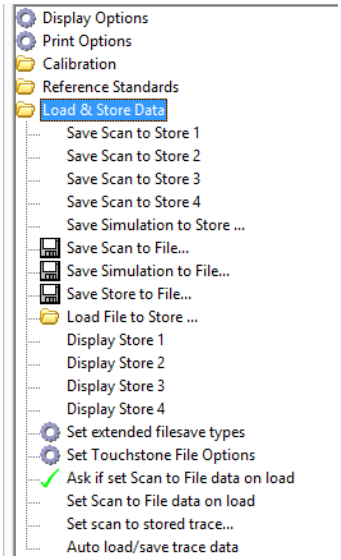
The next 3 values define the Low Loss Capacitor used in LLC calibration. Only the first two parameters are important – the Q and the frequency at which that Q would be expected. The program will use one of three different built in models for how the Q varies with frequency to define the overall characteristics. The model to be used is selected by choosing one of the LLC options towards the bottom of this treeview node. For more information refer to Section 18.4.

The next two options control the use of reference data in calibration files. When calibration is performed, it is performed with a set of reference standards and therefore it makes sense to save the reference standards data with the calibration data and reload it when calibration data is loaded. If you don't want to do this untick one or other of these options.

The final entry is the optional offset of the reference plane. Using the reference standards parameters defined above and a reference standard built carefully from an SMA female, the reference plane will be the back face of the SMA. If you want another reference plane, specify here the offset in millimetres.

35.3.8 Load & Save Treeview Node

This node controls the loading and saving of trace data to internal data stores or to file.


Display Options
Print Options
Calibration
Reference Standards
Load & Store Data
Save Scan to Store 1
Save Scan to Store 2
Save Scan to Store 3
Save Scan to Store 4
Save Simulation to Store ...
Save Scan to File...
Save Simulation to File...
Save Store to File...
Load File to Store ...
Display Store 1
Display Store 2
Display Store 3
Display Store 4
Set extended filesave types
Set Touchstone File Options
Ask if set Scan to File data on load
Set Scan to File data on load
Set scan to stored trace...
Auto load/save trace data

The first four options provide a one click method of saving current scan data into one of the four built in data stores, the next saves simulation data to one of the stores but in this case a dialog box will be used to specify the store to be used (this is to keep the treeeview node to manageable size).

The next three options are used to save scan, simulation or data store trace data to a file and the one following that loads data from file. When data is saved to file, a number of file formats are supported but the available formats will be filtered depending on the instrument mode. For example, Touchstone snp file formats s1p and s2p are supported but will be suppressed if the instrument mode is not set to S parameters.

After the file save & load nodes, the next four are used to display or hide data store trace data. Normally when data is saved to a store or loaded from file to a store, that store is then automatically displayed, but it may be hidden or redisplayed using one of these options.

The next option is used to configure what parameters are saved when a file is saved in "extended csv" format. It permits a variety of parameters to be saved for each data point.

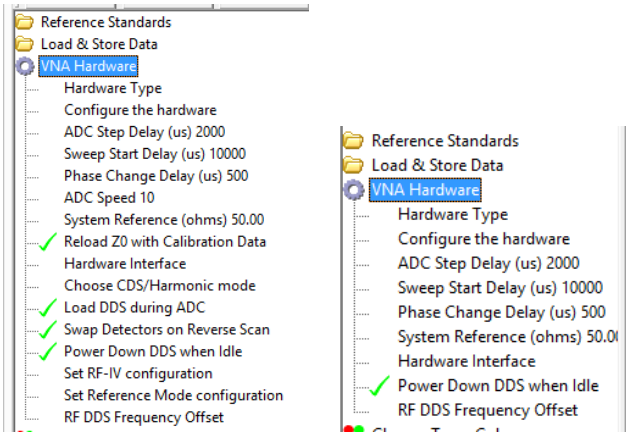
The next option sets the Touchstone file configuration. Data may be saved in S, Y or Z parameter format, and as Magnitude and angle, Magnitude (dB) and angle or real and imaginary.

There then follow two nodes that control behaviour when a file is loaded to a store. The scan can either be automatically set to match the stored data or the user may be prompted as to whether it should be set. Note that this sets the scan, not the display (unless the display is locked to the scan).

In a similar manner to setting the scan to match a trace when loading to a store, the next option permits the scan to be set to match a current data store.

The final option, if ticked, will save the contents of any data stores to temporary files when the program is closed, and will reload any such traces when the program is reopened.

35.3.9 VNA Hardware Treeview Node



This node sets the program configuration to match the hardware. The MSA case presents a reduced set as compared to the N2PK. The N2PK set are shown on the left in the figure above.

Hardware Type specifies the hardware type – N2PK VNA or MSA. Select the desired option.

“Configure the Hardware” is a dialog box whose appearance changes depending on the hardware type. It is used to define the specific details of the hardware. Details on hardware configuration are given in Section 4.2.

The next three entries determine the delays introduced by the program into a sweep. The first is the delay every time a change in DDS frequency to the next step is made before an ADC reading is started. The second is the corresponding delay when the DDSs are set back to the sweep start frequency for a new sweep. The final delay is the one introduced when the LO DDS phase is changed before a reading is made. Whilst software latency can introduce extra delays it should never be faster than the delay entered, and for USB interfaces will be fairly accurate as he delay is managed automatically by the USB interface controller itself.

“ADC Speed” is the DDS conversion time / speed control. It refers to the ADC mode of the LTC2440 ADC chip but it is easier to think of this as speed control with 1 the slowest and 10 the fastest. ADC noise is lower for the slower speeds.

The next two entries concern the system reference impedance. Usually 50 ohms, it may be set as desired, and when calibration is performed, it is stored with the calibration data. The second node, when ticked, causes Z0 to be set from the value read in from the calibration data.

The hardware interface follows; clicking on this node opens the hardware interface dialog. Details may be found in Section 4.3.

Next comes “Choose CDS/Harmonic mode”. This is a digital filter created by Kiciak and Torrey that may be used to implement a range of filtering options. The simplest is Correlated Digital Sampling (CDS) but higher order filters may also be used and it may also be used to select a harmonic of the DDS frequency, extending an N2PK from 60 MHz to 300MHz. It has no effect in MSA mode.

The next three controls affect ADC / DDS operation. Each may be toggled on or off. The first causes the DDS to be loaded for the next reading during an ADC reading. Whilst there is the possibility of introducing some noise as a result of the digital switching, this does not appear to be the case and it speeds up scans. The next causes the detectors to be swapped in their function when a reverse scan is being mode. This is only needed if the specific S parameter test set requires it. Finally, the DDSs may be powered down when not sweeping to save power but beware, the temperature of the chips will then rise when scanning which may cause some thermal drift.

The next two entries configure the software for specific RF-IV or Reference mode test head hardware. The modes themselves are selected from the “Instrument Mode” node of the treeview.

The final entry specifies an optional RF DDS frequency offset that may be applied if selected in the Instrument Mode section.

35.3.10 Choose Trace Colours Treeview Node

This node is used to set the colours of the display. Each opens the same dialog box that allows a colour to be selected. The colour may be set for:

- The trace border
- The graticule
- Traces 1 – 8
- Markers 1 – 9
- The background display colour

"Traces 1-8" needs some explanation. Normally less than 8 traces will be shown at once = it is far too cluttered otherwise. If however more than 8 are shown at once, the colour will wrap around and the 9th trace will use the same pen colour as the 1st

35.3.11 Transverters Treeview Node

The entries in this node vary according to the transverters defined. The first entry opens a dialog box to configure transverters. Each once may be given a name, and each named configured transverter is then shown in the list. Only one may be selected at a time, the default is "none".

35.3.12 Network Settings Treeview Node

There is a simple interface provided that permits a VNA connected to one PC to be used from another PC. There are two entries in this node:

- Configure remote access
- Accept remote access

The first is used on a client PC to tell it how to communicate with another PC that is also running myVNA in server mode. The second entry tells an instance that it is a server and may accept incoming connections.

To configure remote access, the only entry currently needed is the port number. The IP address is specified using the TCP/IP hardware interface option.

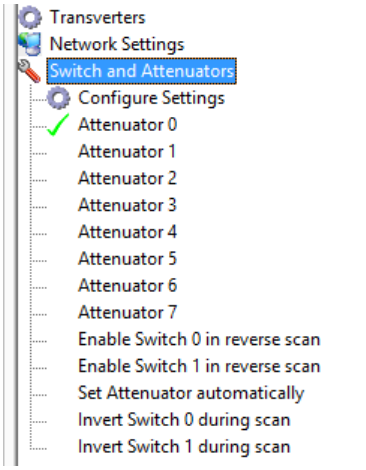
35.3.13 Switches and Attenuators Treeview Node

This node it used to configure the switches and attenuators.

The first entry opens a configuration dialog box that provides the following:

- It permits selection of a specific attenuator setting to be used for forward and (separately) for reverse scans
- It permits attenuator selection nodes to be labelled (or hidden). These labels then appear (or not) in the main Switches and Attenuators Treeview node.
- It permits the output attenuator control lines to be inverted. Normally for example setting 1 has a binary output of '001' but when inverted this appears as '110'.
- Switches may be defined, labelled and either displayed or hidden

The hardware and interface selections made will determine which of these entries are visible or changeable. Be careful in one regard, if you configure settings on one interface (USB) then switch to parallel, the other settings will still exist and be active but will be hidden.



After the "Configure Settings" node, the next set of entries are the attenuators that are configured to be displayed with their names. "Attenuator 0,1,2,3,4,5,6,7" are the default labels given by the program.

The final 5 entries shown above control automatic switch operation. A switch may be set to change state during a reverse scan – this is used to alter the settings of an S parameter test set automatically. The next one causes the attenuator to be set during forward or reverse scans automatically as determined by the settings configured. The final to cause a switch to be inverted when the hardware is actually scanning. It then reverts back at the end or during the delay to the next scan. This may be used as an external trigger for when a sweep is being made.

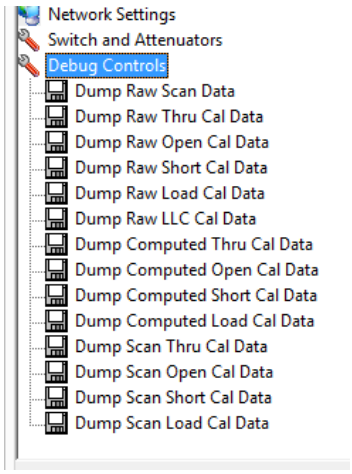
35.3.14 Debug Treeview Node

This node is normally hidden by default, it may be displayed by unticking the "hide debug controls" option. It is not intended for general use.

Each entry in the list provides the facility to save in csv format the:

- Raw scan data
- Raw calibration data (OSLT and LLC)
- Computed calibration data for the scan
- OSLT scan calibration data

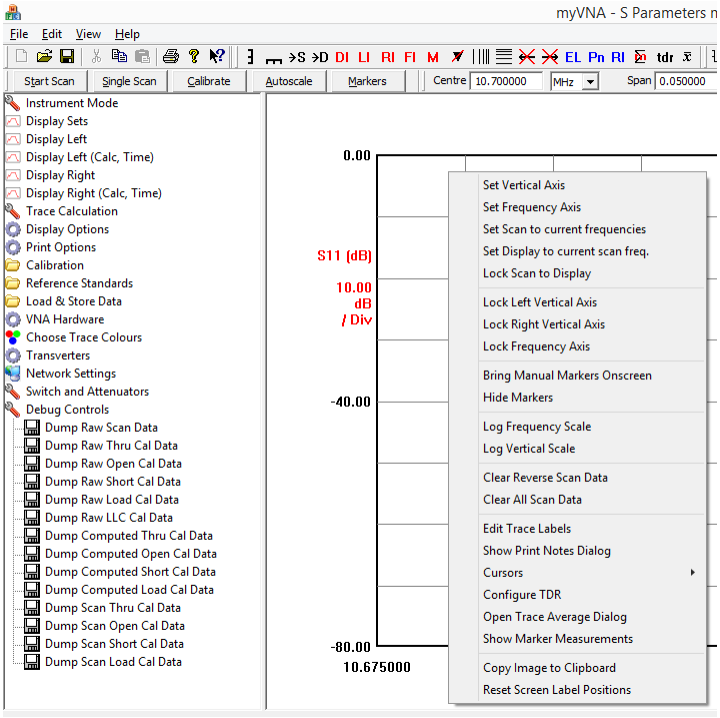
No further details are given here.

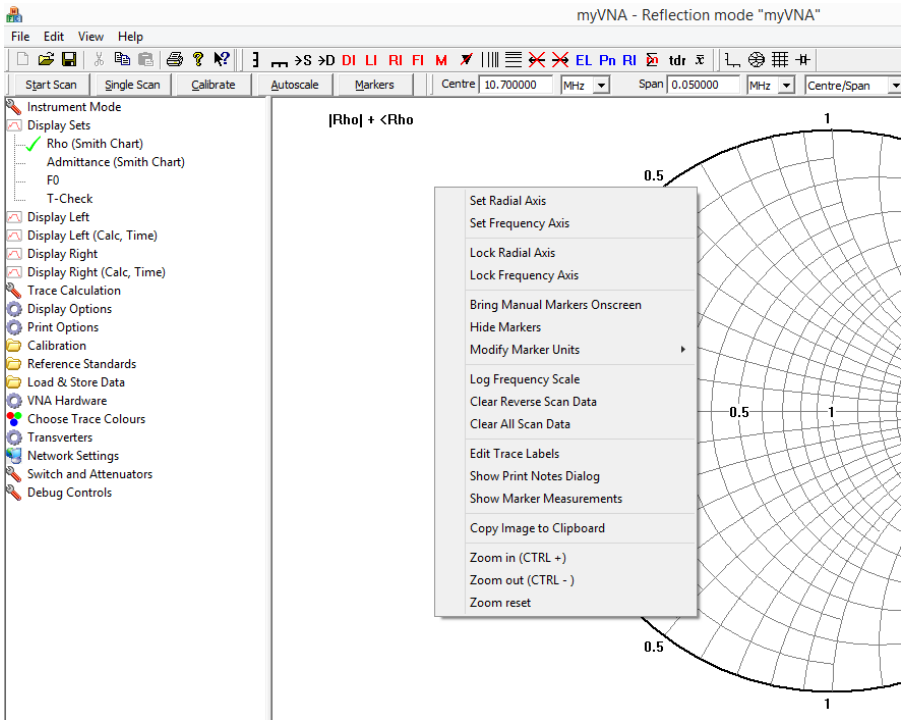

Network Settings
Switch and Attenuators
Debug Controls
Dump Raw Scan Data
Dump Raw Thru Cal Data
Dump Raw Open Cal Data
Dump Raw Short Cal Data
Dump Raw Load Cal Data
Dump Raw LLC Cal Data
Dump Computed Thru Cal Data
Dump Computed Open Cal Data
Dump Computed Short Cal Data
Dump Computed Load Cal Data
Dump Scan Thru Cal Data
Dump Scan Open Cal Data
Dump Scan Short Cal Data
Dump Scan Load Cal Data

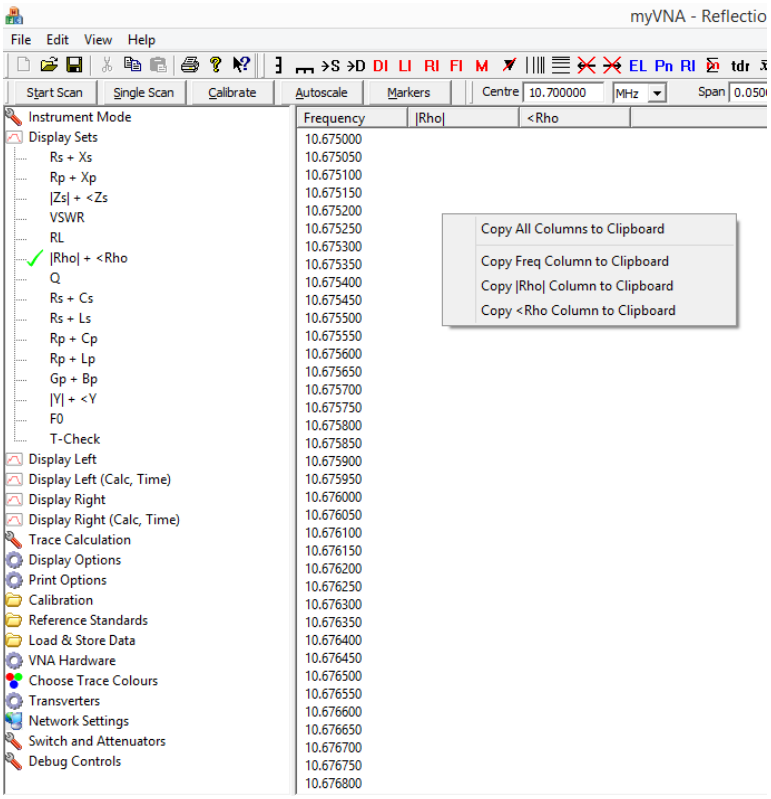
35.4 Popup Menus

There are separate popup menus that appear when right clicking on the diplay area.

Each setting is self evident and they duplicate settings on toolbars or in the treeview controls. Copies of the menus are shown below without further explanation.






myVNA - Reflectio
File Edit View Help
Start Scan
Single Scan
Calibrate
Autoscale
Markers
Centre 10.700000
MHz
Span
Instrument Mode
Display Sets
Rs + Xs
Rp + Xp
|Zs| + <Zs
VSWR
RL
|Rho| + <Rho
Q
Rs + Cs
Rs + Ls
Rp + Cp
Rp + Lp
Gp + Bp
|Y| + <Y
F0
T-Check
Display Left
Display Left (Calc, Time)
Display Right
Display Right (Calc, Time)
Trace Calculation
Display Options
Print Options
Calibration
Reference Standards
Load & Store Data
VNA Hardware
Choose Trace Colours
Transverters
Network Settings
Switch and Attenuators
Debug Controls
Frequency
|Rho|
<Rho
10.675000
10.675050
10.675100
10.675150
10.675200
10.675250
10.675300
10.675350
10.675400
10.675450
10.675500
10.675550
10.675600
10.675650
10.675700
10.675750
10.675800
10.675850
10.675900
10.675950
10.676000
10.676050
10.676100
10.676150
10.676200
10.676250
10.676300
10.676350
10.676400
10.676450
10.676500
10.676550
10.676600
10.676650
10.676700
10.676750
10.676800
Copy All Columns to Clipboard
Copy Freq Column to Clipboard
Copy |Rho| Column to Clipboard
Copy <Rho Column to Clipboard

Annex A - Common Configurations

Can't get a trace on the screen? Check the settings. Here are a simple set to get you going.

Instrument mode:

- Reflection or transmission mode is suggested initially
- Untick 'Perform Reverse Scan'
- Untick 'Use RF-IV mode'
- Untick 'Use Reference Mode'
- Untick 'Always Do Dual Scan'
- Untick 'Logarithmic Scan'
- Untick 'TDR Mode Only'
- Untick 'Apply RF DDS scan freq. offset'
- Make sure "Reflection Data Port" shows "1" as a value
- Set Trace Averages to 1

Display Sets

- Rs+Xs for transmission mode or G(dB)+<G for transmission. Choosing an option under this heading means you can ignore for now Display Left and Display Right

Display Options

- Tick 'Graticule'
- Tick 'Scan Progress Bar'
- Tick 'Autoscale on display change'
- Tick both "snap" options
- Tick "Audio Cues" if you want some beeps
- Tick "Force |display| for log axes
- Untick "Invert RL display"
- Tick "Display Info Tips"
- Set vertical divisions to 8
- Set horizontal divisions to 10
- Set Display Pen Width to 2
- Untick 'Smith/Polar View", "Report View" or "Equivalent Circuits"
- Untick "Auto Refine Eq Cct"
- Tick 'Show Scan Data'
- Set Marker Size to 3

Print Options

- Set Print Pen Width to 5
- Tick 'Label Markers in Printouts'
- Set Marker Size to 5

Calibration

- Untick "Relax harmonic mode checking"
- Tick 'Use Separate ISO Average'
- Set Isolation Average to 5
- Set trace averages to 1
- Untick "Unguided Calibration" initially
- Tick 'Response and OSL Calibration'

- Untick “Log Calibration Scan”
- Untick “Do LLC calibration”
- Untick “Lode mode with Cal data”

Reference Standards

- Set Cop to 0.04
- Set Cld to 0.23
- Set Rld to 50.00
- Set Lld to 0.15
- Set Rsh to 0.00
- Set Lsh to 0.00
- Set Q LLC to 3000
- Set “Q @ Freq (MHz)” to 1.00
- Set C LLC (pF) to 300
- Tick “Fixed Q”
- Tick “Save stds data in cal files”
- Tick “Load stds data with Cal files if present”

Load & Store Data

- Tick “Set scan to file data on load”

VNA Hardware

- Under hardware type select the hardware (N2PK VNA or MSA)
- Under Configure the hardware, for N2PK:
  - If using single detector VNA set both ADCs to 1 otherwise the “usual” configuration is ADC1 for Reflection and ADC2 for transmission.
  - If using dual detector set the ADCs according to you configuration
  - Set clock to 148.343 initially (set it for your hardware later)
  - Set Minimum frequency to 0.06 MHz
  - Set Maximum frequency to 60 MHz
- Under Configure the hardware, for MSA there is no simple default, you need to work it out. Sorry.
- Set ADC step delay to 1000
- Set Sweep Start Delay to 5000
- Set Phase change Delay to 100
- Set ADC Speed to 10 for an N2PK VNA with 2440 ADCs, or 1 for a VNA with 2410s
- Set System reference to 50.00
- From “hardware interface”, choose either USB or Parallel, choose InpOut32 initially and check if using parallel that the port name and base address are correct. Set the parallel port ADC timeout to 500 msec.
- From “Choose CDS/Harmonic mode” set “Harmonic Suppression Mode” and “Fundamental” and “4 samples”
- Tick “Load DDS during ADC”
- Untick “Swap detectors on reverse scan” but remember this needs setting correctly if you use an S parameter test set
- Tick “Power down DDS when idle”

Choose Trace Colours

- Choose suitable colours – not white !

Transverters

- Tick ‘none’

Network Settings

- Nothing to set in this version

Switches and Attenuators

- Select the configuration dialog and ensure all switches are enabled (you may have to temporarily select the USB interface from “Hardware” to see switch1)
- Ensure that for reverse scans, neither switch is ticked in the treeview
- Ensure that the “Set Attenuator Automatically” option is not ticked.

From the “View” menu select “Options” and set the following

- DDS Clock Frequency Limits Min 10, max 10000
- Base VNA frequency limits to 0 (min) and 10000 (max)
- Transverter Frequency Limits 1 (min) and 10000 (max)
- Calibration standard limits 500 (Rld) and 10 (the rest)
- Untick all the “Change LeftView Controls” boxes apart from ‘Hide Debug Controls’ which should be ticked. If any changes were made here, exit the program and restart it to display the missing controls.
- Tick the option “Use Curve fitting for Calibration Data Interpolation”
- Untick the option “Use Log(f) if curve fitting log calibration data”
- Tick the option “Set DDS to scan start freq when waiting for calibration”
- Tick “Permit frequency drag/rescale whilst scanning”
- Set Display frequency limits to 0 and 200
- Tick “save basic mode data in calibration data file”
- Tick “prompt for calibration notes when saving”
- Set lower and upper display limits to 0 and 200 MHz for the N2PK; for the MSA set it according to the hardware.
- Tick “Reset Warnings”

On the main toolbars:

- Select “Full Scan” from the drop down list
- Set Steps to 200
- Set Avg to 1

Now exit the program and restart it. Proceed as per the first time use in the front sections of this manual.

Annex B - S Parameter Test Sets and Bridges

There are basically 3 configurations for the VNA, single detector, dual detector and S parameter test set.

For a single detector configuration, there are basically two configurations; straight through for transmission mode or a single bridge configuration. In the latter case the RF DDS Out and Det RF In ports of the bridge are connected to the corresponding ports on the VNA and the Device Under Test (DUT) connects, not surprisingly to the DUT port. Refer to Figure 11 of Part 2 of Paul's VNA documentation for a circuit diagram. This configuration is shown in Figure A of Paul's Fast Detector publication.

A simple single bridge dual scan configuration is suitable in the following circumstances:

- Dual Scan Response Calibration (RC)
- 5 or 6 term Enhanced Response Calibration (ERC)
- 10 or 12 term calibration with manual device reversal.

In this configuration, the bridge is connected as above, the DUT port on the bridge is connected to the input port of the device under test and the output port is connected to the transmission detector input. When calibrating, the DUT is substituted by an Open Short or Load on the bridge DUT port or by a through connection in Thru mode. This configuration is shown in Figure B of Paul's Fast Detector publication.

An S parameter test set is used with 10 or 12 term calibration with automatic switching. In this configuration two bridges are used with relay switching as shown in Figure C of Paul's fast detector memo. The DUT is connected between the DUT ports of the bridges.

When testing amplifiers it may be advantageous to use different attenuator setting for forward and reverse scans. To support this, the program may be instructed to automatically set an attenuator value before each scan, and to use a different value for forward and reverse scans. To read more on this see the section on Attenuator control.

Annex C - Glitches when using the parallel port

Under XP it is possible to get sudden unexplained glitches such as the one shown below

Whilst signal/cable errors may be the cause, there is also a possible software reason.

Under later versions of XP, the parallel port subsystem will poll for printers automatically; apparently on a periodic interval of 5 seconds and following the detection of activity on the port. This can give rise to glitches like that shown above.

If this happens, you may disable the auto poll function by changing an entry in the registry. Specifically in the following location

```
HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Services\Parport
```

there may be a key called ‘Parameters’. If not, create one and then within that key create an entry called ‘DisableWarmPoll’ of type DWord with a value of ‘1’. Next time you start the computer the problem should have gone away but it is worth checking with RegEdit that it is still present.

Note also that a search on the Internet for ‘DisableWarmPoll’ will solicit many similar postings; this is not a unique problem. However guidance seems to vary. Some say to put the entry into ControlSet0001, some into a key ‘Parameters’ within that location. I have found that if you do this, the entry will be erased on startup.

To re-enable the warm polling, set the value to ‘0’.

To do this automatically, create a couple of text files called ‘EnableWarmPoll.reg’ and ‘DisableWarmPoll.reg’ with settings of 0 and 1 respectively. The contents of the files would be as shown below. Note that for myVNA, files like this will be

automatically installed in the program installation directory under a sub directory called ‘Parallel Port Configuration’

For DisableWarmPoll.reg:

```
Windows Registry Editor Version 5.00

[HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Services\Parport\parameters]
"DisableWarmPoll"=dword:00000001
```

And for EnableWarmPoll.reg:

```
Windows Registry Editor Version 5.00

[HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Services\Parport\parameters]
"DisableWarmPoll"=dword:00000000
```

Double click on the appropriate file. Two confirmation dialogs will be displayed, one to check you want to modify the registry and one to confirm the change.

If you find this does not work or that the key needs to be elsewhere to work on your system, please let me know.

Annex D - Automation interface example contents

```
// myVNA.odl : type library source for myVNA.exe

// This file will be processed by the Make Type Library (mktyplib) tool to
// produce the type library (myVNA.tlb).

[ uuid(AAC33E4F-AB38-4100-BF4A-C779F89AAC58), version(1.3) ]
library myVNA
{
    importlib("stdole32.tlb");

    //  Primary dispatch interface for CmyVNADoc

    [ uuid(1D00566F-4A33-4bcf-AB14-A73EAF0006CE) ]
    dispinterface IID_IA_myVNA
    {
        properties:
            // NOTE - ClassWizard will maintain property information here.
            //    Use extreme caution when editing this section.
            //{{AFX_ODL_PROP(CmyVNADoc)
            [id(1)] int nScanSteps;
            [id(3)] int nScanAverage;
            [id(9)] int nInstrumentMode;
            [id(19)] int nDisplayMode;
            //}}AFX_ODL_PROP

        methods:
            // NOTE - ClassWizard will maintain method information here.
            //    Use extreme caution when editing this section.
            //{{AFX_ODL_METHOD(CmyVNADoc)
            [id(2), helpstring("method: Do a single scan")] int SingleScanAutomation(LONG Message, HWND hWnd, LONG Command, LONG lParam );
            [id(4), helpstring("method: Set current frequency range for scan")] int SetScanDetailsAutomation(double dF1, double dF2, LONG nFlags);
            [id(5), helpstring("method: generic method to get double data to an array")] int GetDoubleArrayAutomation(LONG nWhat, LONG nIndex,
LONG nSize, VARIANT *a);
            [id(6), helpstring("method: generic method to set double data from an array")] int SetDoubleArrayAutomation(LONG nWhat, LONG nIndex,
LONG nSize, VARIANT *a);
            [id(7), helpstring("method: Show or hide myVNA window")] void ShowWindowAutomation(LONG nValue);
            [id(8), helpstring("method: Retrieve subset of current scan data results")] int GetScanDataAutomation(LONG nStart, LONG nEnd, LONG
nWhata, LONG nWhatb, VARIANT *a, VARIANT *b );
            [id(10), helpstring("method: Load current configuration data from a file")] int LoadConfigurationAutomation(LPCTSTR fileName);
            [id(11), helpstring("method: Save current configuration data to a file")] int SaveConfigurationAutomation(LPCTSTR fileName);
```

```
            [id(12), helpstring("method: Load current calibration data from a file")] int LoadCalibrationAutomation(LPCTSTR fileName);
            [id(13), helpstring("method: Save current calibration data to a file")] int SaveCalibrationAutomation(LPCTSTR fileName);
            [id(14), helpstring("method: generic method to get integer data to an array")] int GetIntegerArrayAutomation(LONG nWhat, LONG nIndex,
LONG nSize, VARIANT *a);
            [id(15), helpstring("method: generic method to set integer data from an array")] int SetIntegerArrayAutomation(LONG nWhat, LONG
nIndex, LONG nSize, VARIANT *a);
            [id(16), helpstring("method: Autoscale display")] int AutoscaleAutomation(void);
            [id(17), helpstring("method: Copy current displayed data to clipboard")] int ClipboardCopyAutomation(void);
            [id(18), helpstring("method: Save current trace data to a file")] int SaveTraceDataAutomation(LPCTSTR fileName);
            [id(20), helpstring("method: Refine Equivalent Circuit")] int RefineAutomation([in] LONG Message, [in] LONG hWnd, [in] LONG Command,
[in] LONG lParam);
            [id(21), helpstring("method: Select Equivalent Circuit log file")] int SetEqCctLogFileAutomation(LPCTSTR fileName);
            [id(22), helpstring("method: Add Equivalent Circuit results to log file")] int EqCctLogFileAutomation(LPCTSTR description);
            [id(23), helpstring("method: Set a string parameter")] int SetStringAutomation(LONG nWhat, LONG nIndex, LPCTSTR description);
            [id(24), helpstring("method: Get a string parameter")] BSTR GetStringAutomation(LONG nWhat, LONG nIndex );
            //}}AFX_ODL_METHOD

    };

    //  Class information for CmyVNADoc

    [ uuid(4D9364D3-3187-4cea-A28D-5DC06E343344) ]
    coclass CmyVNADoc
    {
        [default] dispinterface IID_IA_myVNA;
    };


    //{{AFX_APPEND_ODL}}
};
```

The following is an extract from AccessMyVNA.h.
The functions referenced are the simple ones in AccessMyVNA.cpp, but there is a 1:1 correspondence to the OLE interface. The corresponding OLE calls are indicated for each

```
//
////////////////////////////////////////////////////////////////////////////////////////
//
// MyVNAInit()
// call this function befoe trying to do anything else. It attempts to execute myVNA
// and establish it as an automation server
// OLE equivalent:
// Connect to the server using the CLSID
__declspec(dllexport) int _stdcall MyVNAInit(void);

// Make sure you call the Close function before the windows GUI application is closed
// OLE equivalent:
// simply release the interface
__declspec(dllexport) int _stdcall MyVNAClose(void);


//
////////////////////////////////////////////////////////////////////////////////////////
//
// MyVNASingleScan()
// attempt to single scan the VNA. On completion myVNA will post a message ( Message) to the queue
of the specified
// window ( hWnd ) with the given COmmand and lParam values. See the example in AccessMyVNADlg.cpp
// OLE equivalent:
// int SingleScanAutomation(LONG Message, HWND hWnd, LONG lParam, LONG wParam );
__declspec(dllexport) int _stdcall MyVNASingleScan(int Message, HWND hWnd, int nCommand, int lParam
);

//
////////////////////////////////////////////////////////////////////////////////////////
//
// MyVNAEqCctRefine()
// attempt to refine an equivalent circuit. On completion myVNA will post a message ( Message) to
the queue of the specified
// window ( hWnd ) with the given Command and lParam values. See the example in AccessMyVNADlg.cpp
// OLE equivalent:
// int RefineAutomation(LONG nEnd, LONG hWnd, LONG Command, LONG lParam);
__declspec(dllexport) int _stdcall MyVNAEqCctRefine(int Message, HWND hWnd, int nCommand, int
lParam );


//
////////////////////////////////////////////////////////////////////////////////////////
//
// MyVNASetScanSteps()
// sets the number of steps ( min 2 max 50,000 but note this can take a huge amount of memory)
// OLE equivalent:
// Set property nScanSteps
__declspec(dllexport) int _stdcall MyVNASetScanSteps(int nSteps);

//
////////////////////////////////////////////////////////////////////////////////////////
//
// MyVNAGetScanSteps()
// read the current scan steps value
// OLE equivalent:
// Get property nScanSteps
__declspec(dllexport) int _stdcall MyVNAGetScanSteps(int *pnSteps);

//
////////////////////////////////////////////////////////////////////////////////////////
//
// MyVNASetScanAverage() and MyVNAGetScanAverage()
// set / get the current scan trace average count
// OLE equivalent:
// Set / Get property nAverage
__declspec(dllexport) int _stdcall MyVNASetScanAverage(int nAverage);
__declspec(dllexport) int _stdcall MyVNAGetScanAverage(int *pnAverage);
```

```
//
////////////////////////////////////////////////////////////////////////////////////////////
//
// MyVNASetFequencies
// nFlags bits 0..3 set scan mode where f1 is first parameter / f2 is the second
// 0 = centre / span
// 1 = start / stop
// 2 = full scan
// 3 = from zero
// 4 = centre / per division
// 5 = centre / per step
// 6 = start / per step
// 7 = start / per division
// 8..15 unused
//
// return codes:
// 0 - OK
// 1 - start too low
// 2 - stop too high
// 3 - width must be > 0
// 4 - start is before end
// 5 - stop too low
// OLE equivalent:
// int SetScanDetailsAutomation(double dF1, double dF2, LONG nFlags);
__declspec(dllexport) int _stdcall MyVNASetFequencies(double f1, double f2, int nFlags);

//
////////////////////////////////////////////////////////////////////////////////////////////
//
// MyVNAShowWindow()
// show or hide the current myVNA window.
// nValue set to 0 causes SW_SHOWDEFAULT
// nValue set to 1 causes SW_SHOWMINIMIZED
// all other values ignored
// OLE equivalent:
// void ShowWindowAutomation(LONG nValue);
__declspec(dllexport) int _stdcall MyVNAShowWindow(int nValue);

//
////////////////////////////////////////////////////////////////////////////////////////////
//
// MyVNAGetScanData()
// get current scan data results starting from scan point nStart up to and including nEnd
// note that a scan of N steps gives N+1 data points from 0 to N
// the data type requsted( nWhat) must be one of the ones as follows.
// It is the callers' responsibility
// to make sure that that data type is capable of being calculatd from the current scan data
//
// OLE equivalent:
// int GetScanDataAutomation(LONG nStart, LONG nEnd, LONG nWhata, LONG nWhatb, VARIANT *a, VARIANT
*b );
// VARIANT a and b must encapsulate safearrays of doubles of sufficient size for the output and be
zero indexed
// even if one or other ( a or b ) are not in use - i.e. DISPLAY_NOTHING is used for it, then an
suitable
// variant must be provided still
__declspec(dllexport) int _stdcall MyVNAGetScanData(int nStart, int nEnd, int nWhata, int nWhatb,
double *pDataA, double *pDataB );
// flags for nWhat in GetScanData
// the first is a dummy - used for nWhata or nWHatb to cause no data to be retrieved for that
// case ( a or b) hence to retrieve just one parameter set in a call, set nWhatA or nWhatB to the
// desired value and set the other (nWhatb or nWhata) to be set to -2.
// Otherwise two separate parameter values may be retrieved at same time, for example setting
// nWhata to -1 and nWhatb to 0 would cause scan frequency data and RS data to be retrieved.
// Setting the values to 21 and 22 would cause S11 real and imaginary to be retrieved.
#define DISPLAY_NOTHING -2
#define DISPLAY_FREQ_SCALE -1
#define DISPLAY_REFL_RS 0
#define DISPLAY_REFL_XS 1
#define DISPLAY_REFL_RP 2
#define DISPLAY_REFL_XP 3
#define DISPLAY_REFL_MODZS 4
#define DISPLAY_REFL_ANGZS 5
#define DISPLAY_REFL_VSWR 6
#define DISPLAY_REFL_RL 7
#define DISPLAY_REFL_RHO 8
#define DISPLAY_REFL_ANGRHO 9
```

```
#define DISPLAY_REFL_Q 10
#define DISPLAY_REFL_CS 11
#define DISPLAY_REFL_LS 12
#define DISPLAY_REFL_CP 13
#define DISPLAY_REFL_LP 14
#define DISPLAY_REFL_GP 15
#define DISPLAY_REFL_BP 16
#define DISPLAY_REFL_MODY 17
#define DISPLAY_REFL_ANGY 18
#define DISPLAY_REFL_RHO_REAL 19
#define DISPLAY_REFL_RHO_IMAG 20
#define DISPLAY_S11_REAL 21
#define DISPLAY_S11_IMAG 22
#define DISPLAY_S11_ABS 23
#define DISPLAY_S11_ANG 24
#define DISPLAY_S11_RL 25
#define DISPLAY_S21_GAIN 26
#define DISPLAY_S21_GAIN_DB 27
#define DISPLAY_S21_GAIN_ANG 28
#define DISPLAY_S22_REAL 29
#define DISPLAY_S22_IMAG 30
#define DISPLAY_S22_ABS 31
#define DISPLAY_S22_ANG 32
#define DISPLAY_S22_RL 33
#define DISPLAY_S12_GAIN 34
#define DISPLAY_S12_GAIN_DB 35
#define DISPLAY_S12_GAIN_ANG 36
#define DISPLAY_S21_REAL 37
#define DISPLAY_S21_IMAG 38
#define DISPLAY_S12_REAL 39
#define DISPLAY_S12_IMAG 40
#define DISPLAY_TRANS_GAIN 41
#define DISPLAY_TRANS_GAINDB 42
#define DISPLAY_TRANS_GAIN_PHASE 43
#define DISPLAY_TRANS_REAL 44
#define DISPLAY_TRANS_IMAG 45
#define DISPLAY_TRANS_GROUPDELAY 46
#define DISPLAY_DISP1 47
#define DISPLAY_DISP1RE DISPLAY_DISP1
#define DISPLAY_DISP1IM DISPLAY_DISP1+1
#define DISPLAY_DISP2 49
#define DISPLAY_DISP2RE DISPLAY_DISP2
#define DISPLAY_DISP2IM DISPLAY_DISP2+1
#define DISPLAY_DISP3 51
#define DISPLAY_DISP3RE DISPLAY_DISP3
#define DISPLAY_DISP3IM DISPLAY_DISP3+1
#define DISPLAY_DISP4 53
#define DISPLAY_DISP4RE DISPLAY_DISP4
#define DISPLAY_DISP4IM DISPLAY_DISP4+1
// spectrum analyser trace - unused
#define DISPLAY_SPECTRUM_MAG 55
#define DISPLAY_SPECTRUM_DBM 56
// time domain - offsets in nDisplayTDRType
#define DISPLAY_TDR_FIRST 57
#define DISPLAY_TDR_11_V 57
#define DISPLAY_TDR_11_DB 58
#define DISPLAY_TDR_22_V 59
#define DISPLAY_TDR_22_DB 60
#define DISPLAY_TDR_21_V 61
#define DISPLAY_TDR_21_DB 62
#define DISPLAY_TDR_12_V 63
#define DISPLAY_TDR_12_DB 64
#define DISPLAY_TDR_11_ZS 65
#define DISPLAY_TDR_22_ZS 66

//
////////////////////////////////////////////////////////////////////////////////////////////////////
//
// MyVNASetDisplayMode() and MyVNAGetDisplayMode()
// get and set the basic display mode
__declspec(dllexport) int _stdcall MyVNASetDisplayMode(int nMode);
__declspec(dllexport) int _stdcall MyVNAGetDisplayMode(int *pnMode);
// nMode takes one of these values
#define DISP_MODE_RECT 0
#define DISP_MODE_REPORT 1
#define DISP_MODE_EQ_CCT 2
#define DISP_MODE_POLAR 3
```

```
//
////////////////////////////////////////////////////////////////////////////////////////////
//
// MyVNASetInstrumentMode() and MyVNAGetInstrumentMode()
// get and set the basic instrument mode
// OLE equivalent:
// Set / Get property nInstrumentMode
__declspec(dllexport) int _stdcall MyVNASetInstrumentMode(int nMode);
__declspec(dllexport) int _stdcall MyVNAGetInstrumentMode(int *pnMode);
// data structure for Get/Set instrument mode
// parameter is a 32 bit unsigned integer as follows
// bits 0..3 define the mode. Not all values currently in use
// the following values should be placed into those bits as follows
#define INSTRUMENT_MODE_REFLECTION 0
#define INSTRUMENT_MODE_TRANSMISSION 1
#define INSTRUMENT_MODE_DUALMODE 2
#define INSTRUMENT_MODE_SPARAM 3
#define INSTRUMENT_MODE_SPECANAL 4
// bit 4 if set causes the program to always do a dual scan even if reflection or transmission mode
set
#define ALWAYS_DO_DUAL_SCAN
// bit 5 if set forces a reverse scan (as in S12/S22 instead of S21/S11
#define REVERSE_SCAN (1<<5)
// bit 6 if set causes RFIV mode to be selected
#define RFIV_SCAN (1<<6)
// bit 7 if set causes Reference mode to be selected
#define REFMODE_SCAN (1<<7)
// bit 8 if set causes log frequency scale mode to be selected
#define LOGF_SCAN (1<<8)

//
////////////////////////////////////////////////////////////////////////////////////////////
//
// MyVNALoadConfiguration()
// MyVNASaveConfiguration()
// MyVNALoadCalibration()
// MyVNASaveCalibration()
// Given a filename, attempt to load or save the current program configuration
// including calibration data or just load/save the calibration data
// finally, option exists to save current trace data to a file ( s2p only supported here)
// OLE equivalents:
// int LoadConfigurationAutomation(LPCTSTR fileName);
// int SaveConfigurationAutomation(LPCTSTR fileName);
// int LoadCalibrationAutomation(LPCTSTR fileName);
// int SaveCalibrationAutomation(LPCTSTR fileName);
// int SaveTraceDataAutomation(LPCTSTR fileName);
__declspec(dllexport) int _stdcall MyVNALoadConfiguration( _TCHAR * fileName);
__declspec(dllexport) int _stdcall MyVNASaveConfiguration( _TCHAR * fileName);
__declspec(dllexport) int _stdcall MyVNALoadCalibration( _TCHAR * fileName);
__declspec(dllexport) int _stdcall MyVNASaveCalibration( _TCHAR * fileName);
__declspec(dllexport) int _stdcall MyVNASaveTraceData( _TCHAR * fileName);

//
////////////////////////////////////////////////////////////////////////////////////////////
//
// when in eq cct view mode (and only then) use these functions to establish a log file and log
// the results of a scan. The description is a string added to each log entry
// OLE equivalents:
// int SetEqCctLogFileAutomation(LPCTSTR fileName);
// int EqCctLogFileAutomation(LPCTSTR description);
__declspec(dllexport) int _stdcall MyVNASetEqCctLogFile(_TCHAR * fileName);
__declspec(dllexport) int _stdcall MyVNALogEqCctResults(_TCHAR * description);


//
////////////////////////////////////////////////////////////////////////////////////////////
//
// MyVNAAutoscale()
// execute the autoscale function (same as clicking the Autoscale button)
// OLE equivalent:
// int AutoscaleAutomation(void);
__declspec(dllexport) int _stdcall MyVNAAutoscale(void);

//
////////////////////////////////////////////////////////////////////////////////////////////
//
```

```
// copy whatever is currently being displayed to the clipboard
// OLE equivalent:
// int ClipboardCopyAutomation(void);
__declspec(dllexport) int _stdcall MyVNAClipboardCopy(void);

//
////////////////////////////////////////////////////////////////////////////////////////////
//
// general purpose interface functions used to get or set various things.
// Two versions exist, one for integers and one for doubles, with a Get and a Set in each case
// The functions need a parameter to say what is to be set/got - see details below, a pointer
// to an array of sufficient size for the results and as a safeguard the number of entries
// in tht array
// OLE equivalents:
// int GetIntegerArrayAutomation(LONG nWhat, LONG nIndex, LONG nSize, VARIANT *a);
// int SetIntegerArrayAutomation(LONG nWhat, LONG nIndex, LONG nSize, VARIANT *a);
__declspec(dllexport) int _stdcall MyVNAGetIntegerArray(int nWhat, int nIndex, int nArraySize, int
*pnResult);
__declspec(dllexport) int _stdcall MyVNASetIntegerArray(int nWhat, int nIndex, int nArraySize, int
*pnData);
// options for nWhat parameter in MyVNAGetIntegerArray() and MyVNASetIntegerArray()
// all get / set an array of integers
// nIndex is required for some but not all options - as indicated below. Set to 0 when not used.
//
// case 0 - display options - array of 4 integers
// nIndex not used- set to 0
// data[0] = horizontal divisions in byte 0, vertical divisions in byte 1
// data[1] = byte 0 pen width, byte 1 marker size
// data[2] = flags as follows
//     bit 0 - graticule on
//     bit 1 - scan progress bar displayed
//     bit 2 - autoscale on display change
//     bit 3 - snap to 125 on display change
//     bit 4 - snap to 125
//     bit 5 - audio cues
//     bit 6 - force |disp| on log axes
//     bit 7 - auto refine on equivalent circuits
//     bit 8 - invert RL display
//     bit 9 - display info tips
//     bit 10 - label frequency gridlines
//     bit 11 - label vertical gridlines
//     bit 12 - show scan data
//     bit 13 - lock scan to display
//      bit 14 - 31 spare
// data[3] = more flags. Note these flags are not readable - they will always read as zero in this
version
// this parameter may be omitted by setting the number of integers to 3 instead of 4.
//     bit 0 - log vertical scale if currently in rectangular display mode
//     bit 1 - log frequency scale if currently in rectangular display mode
//     bit 2 - if set, set the scan to match the current display freqeuncies (make sure scan is not
locked to display - see bit 13 above)
//     bit 3 - if set, set the display to match the current scan frequencies (make sure scan is not
locked to display - see bit 13 above)
//     bit 4 - if set lock the frequency axis
//     bit 5 - if set lock the left axis
//     bit 6 - if set lock the right axis
#define GETSET_DISPLAY_OPTIONS 0

// case 1 - print options - array of 4 integers
// nIndex not used- set to 0
// data[0] = unused
// data[1] = byte 0 pen width, byte 1 marker size
// data[2] = flags as follows
//     bit 0 - add print notes to clipboard copy
//     bit 1 - label markers in printout
//     bit 2 - 31 spare
// data[3] = spare
#define GETSET_PRINT_OPTIONS 1

// case 2 - get and set screen colours - array of 1 integers
// nIndex - which colour to access.
//     0 = Border
//     1 = graticule
//     0x10 to 0x17 = trace colours 1 to 8
//     0x30 to 0x38 = marker colours 1 to 9
// data[0] - the colour. This is a DWORD passed as an int
// it is a COLORREF, which takes the form 0x00bbggrr
```

```
// and may be created with the RGB macro.
// to set a colour, populate data[0] and data[1] with the appropriate values
// to get a colour, populate data[0] with the desired target and the subroutine
// will fill in the colour value in data[1]
// if the value of data[0] is out of range, the set command has no effect
// if it is out of range on a get, the returned colour value will be -1 and the subroutine
// will return a non zero value
#define GETSET_COLOURS 2

// case 3 - get or set left axis display parameters
// case 4 - get or set right axis display parameters
// nIndex not used- set to 0
// requires an array of 4 integers
// The array contents correstpond to a 128 bit bitmap
// where the bits that are set determine which parameters are shown on the axis
// The bits correspond to the values shown for Get or Set Scan Data above
// with the exception of DISPLAY_FREQ_SCALE
// hence for example:
// DISPLAY_REFL_CS takes the value of 11 and DISPLAY_REFL_XS takes the value 1
// so the bitmap would be (1<<DISPLAY_REFL_XS) + (1<<DISPLAY_REFL_CS)
// in other words (1<<11) + (1<<1) or 0x00000000000000000000000000000802
// the value is split into four 32 bit unsigned integer values as follows
// data[0] - bits 31-0
// data[1] - bits 63-32
// data[2] = bits 95-64
// data[3] - bits 127-96
// Given the current definitions the final entry in the above list is DISPLAY_TDR_22_ZS
// which takes the value of 66, hence currently bits 67-127 will always be zero.
#define GETSETLEFTAXIS 3
#define GETSETRIGHTAXIS 4

// case 5 - get or set hardware delays & adc speed
// nIndex not used- set to 0
// data[0] - ADC speed
// data[1] - ADC step delay
// data[2] - sweep start delay
// data[3] - phase change delay
// the speed is an integer that depends on hardware; 1..10 for the N2PK other values ignored
// the time delays are integers in us
// the data may be truncated; for example setting the data length to 2 will changeADC speed and
step delay only
#define GETSETHARDWARESPEEDS 5

// case 6 - set or get the hardware options
// nIndex not used- set to 0
// data[0] - CDS mode as defined below
// data[1] - flags as defined below
// data[2] - system reference
// CDS mode. This is an integer structured as follows
// bit 0 - if 0, basic mode and rest of this integer has no effect
//       - if 1, harmonic suppression mode as defined below
// bits 8-15 - harmonic where 0x01 means fundamental, 0x02 is second harmonic
// bits 16-24 - number of samples 0x01 means 1, 0x04 means 4 etc
// limitations:
// samples must be 0x04, 0x08, 0x10 or 0x32. Other values will cause setting to be ignored
// harmonics must be 1,2,3,4 or 5. Other values will be ignored
// if harmonics is 2 or 3, sample setting 4 is not available
// if harmonics is 4 or 5, sample settings 4 and 8 are not available
// example
// to select harmonic mode 3 with 16 samples data[0] should be set to 0x00100301
// it is acceptable to issue the command with just one integer and change the CDS mode without
updating the rest
// data[1] - flags
// bit 0 - if set load DDS during ADC setting
// bit 1 - if set swap detectors on reverse scan
// bit 2 - if set power down DDS when idle
// it is permitted to set the CDS mode and flags without changing system reference by passing just
2 integers
// data[2] - system reference (milli ohms) - must be > 0
#define GETSETHARDWAREOPTIONS 6

// case 7 - set or get marker configuration.
// note there are other functions to get the marker value / frequency and marker arithmetic and
other settings
// nIndex - set to the marker number ( 0..(NUM_MARKERS-1) )
// function will use data[0] to determine which marker and will fill in results in array
// data[0] - source information
```

```
// data[1] - mode information
// data[2] - target
// data[3] - link
// data[4] - display flag
// the meaning of the above is as follows
//
// Source is as follows
// results if set to invalid settings are undefined.
// for example do not set more than 1 of bits 1-3
// do not set bits 8-11 out of range
// do not set bits 16-23 to invalid value ( 0..66 are valid at moment)
// bit 0; 0=>left, 1=>right
// bit 1; 1=>scan data
// bit 2; 1=>store data
// bit 3; 1=>sim data
// bit 4-7 spare
// bit 11-8; store, sim or cal index (0..15)
// bit 15-12 spare
// bit 23-16 parameter type ( not all in use - integer 0..255 )
// bits 31-24 spare
//
// mode is as follows:
// 0 = tracking
// 1 = manual
// 2 = linked
// 3 = linked; f-
// 4 = linked; f+
//
// target is as follows
// 0 = maximum
// 1 = minimum
// 2 = cross up 1st
// 3 = cross down 1st
// 4 = cross up 2nd
// 5 = cross down 2nd
// 6 = cross up 3rd
// 7 = cross down 3rd
//
// link is the other marker number ( 0 .. (NUM_MARKERS-1) )
// display flag is 0 to disable/hide and <>0 to enable & display
#define GETSETMARKERSETUP 7

// case 8 - get (not set) various program constants
// nIndex = 0 - return the following
// data[0] - number of different left/right axis parameters
// data[1] - number of markers
// data[2] - number of calculation markers
// data[3] - number of stores for traces
// data[4] - number of annotations
// data[5] - number of separate trace colour
// data[6] - number of transverters permitted
// data[7] - limit on length of a transverter name
// data[8] - number of simulations supported
// data[9] - number of simulation structures per simulation supported
// data[10] - number of components per simulation
#define GETPROGRAMCONSTANTS 8

// case 9 - get or set equivalent circuit configuration
// nIndex not used - set to 0
// Note: this function is only supported when equivalent circuit display mode is selected
// and unless default options are desired MUST be sent each time the display mode is set to eq cct
mode
// data[0] - equivalent circuit device type. Set to 0 for crystal motional parameters
// data[1] - model - set as follows
//         0 = 45 degree phase
//         1 = 3dB
//         2 = 6 term
// data[2] - set data source
//         0 = current scan data
//         1..number of stores = stored trace data
#define GETSETEQCCTCONFIG 9

// case 10 - get or set simulation configuration
// nIndex not used - set to 0
// sets overall configuration by determining the type of each block
// data[0] is simulation block 1
// data[1] is block 1 etc
```

```
// each one takes values thus
// 0 = unused
// 1 = simulation
// 2 = scan data
// 3 = store 1
// 4 = store 2
// etc
#define GETSETSIMULATIONCONFIG 10

// case 11 - trace calculation options
// nIndex not used - set to 0
// sets options related to trace calculation controls
// data[0] are various flags
//     bit 0 - if set show network simulation dialog
//     bit 1 - if set, set "show simulation data"
//     bit 2 - if set show marker measurements dialog
//     bit 3 - if set, set "TDR functions"
// when read, bits 0 and 2 will always read '0'
#define GETSETTRACECALCULATIONOPTIONS 11

// case 12 - switch and attenuator configuration
// nIndex not used - set to 0
// sets options related to configuration of switch and attenuator options
// data[0] are various flags
//     bit 0 - if set set "invert sense" flag for switch 1
//     bit 1 - if set set "invert sense" flag for switch 2
//     bit 8 - if set set "invert sense" flag for attenuator
// data[1] - values 0 to 7 configure the "forward scan" attenuator setting
// data[2] - values 0 to 7 configure the "reverse scan" attenuator setting
#define GETSETSWITCHATTENUATORCONFIG 12

// case 13 - switch and attenuator settings
// nIndex not used - set to 0
// sets options related to configuration of switch and attenuator settings
// data[0] are various flags
//     bit 0 - if set set "invert sense" flag for switch 1
//     bit 1 - if set set "invert sense" flag for switch 2
//     bit 8 - if set set enable switch 1 during reverse scan
//     bit 9 - if set set enable switch 2 during reverse scan
//     bit 16 - if set set enable switch 1 during scan
//     bit 17 - if set set enable switch 2 during scan
//     bit 24 - if set set enable automatic attenuator setting during scan
// data[1] - values 0 to 7 sets the attenuator
#define GETSETSWITCHATTENUATORSETTINGS 13


//
////////////////////////////////////////////////////////////////////////////////////////////
//
// general purpose interface functions used to get or set various things.
// Two versions exist, one for integers and one for doubles, with a Get and a Set in each case
// The functions need a parameter to say what is to be set/got - see details below, a pointer
// to an array of sufficient size for the results and as a safeguard the number of entries
// in tht array
// OLE equivalents:
// int GetDoubleArrayAutomation(LONG nWhat, LONG nIndex, LONG nSize, VARIANT *a);
// int SetDoubleArrayAutomation(LONG nWhat, LONG nIndex, LONG nSize, VARIANT *a);
__declspec(dllexport) int _stdcall MyVNAGetDoubleArray(int nWhat, int nIndex, int nArraySize,
double *pnResult);
__declspec(dllexport) int _stdcall MyVNASetDoubleArray(int nWhat, int nIndex, int nArraySize,
double *pnData);
//
// options for MyVNAGetDoubleArray and MyVNASetDoubleArray
// case 0 - get (not set) frequency data
// nIndex is not used - set to 0
//  returns a set of doubles for scan related frequency data as follows
//  data[0] = actual scan frequency start
//  data[1] = actual scan frequency end
//  data[2] = scan start (for options that use a start frequency)
//  data[3] = scan end (for options that use a start frequency)
//  data[4] = scan centre (for options that use a centre frequency)
//  data[5] = scan span (for options that use a span frequency)
//  data[6] = scan freq/step (for options that use a freq/step frequency)
//  data[7] = scan freq/division (for options that use a freq/division frequency)
//  data[8] = current scan mode (an integer as defined above returned as a double)
#define GET_SCAN_FREQ_DATA 0
```

```
// case 1 - get or set marker values
// passes an array of values as follows
// nIndex = marker number 0..(NUM_MARKERS-1)
// data[0] = marker value
// data[1] = time or frequency value
#define GETSET_MARKER_VALUES 1

// case 2 - get (not set) equivalent circuit results
// nIndex is not used
// data depends on model chosen
#define GET_EQCCT_VALUES 2

// case 3 - get/set display frequency and time settings
// nIndex is not used
// data[0] - display start frequency
// data[1] - display end frequency
// data[2] - display start time
// data[3] - display end time
// Either 2 or 4 values may be provided, either frequencies alone or frequencies and tim
#define GETSET_DISPLAY_FT_AXIS_VALUES 3

// case 4 - get/set vertical axis settings
// nIndex is 0 for left, 1 for right axis
// data[0] - axis top
// data[1] - axis bottom
#define GETSET_DISPLAY_VERTICAL_AXIS_VALUES 4

// case 5 - get/set N2PK hardware configuration
// nIndex unused
// data[0] - transmission ADV 1 or 2
// data[1] - reflection ADV 1 or 2
// may either pass 2 values or 5; if 2 only the above are affected
// data[2] - clock frequency (Hz)
// data[3] - VNA minimum frequency
// data[4] - VNA maximum frequency
#define GETSET_N2PK_HARDWARE_VALUES 5

// case 6 - get/set network simulation components
// nIndex - simulation number ( 0..{num_simulations-1} )
// data[0] - component 0 type (0=none; 1=R, 2=L, 3=C)
// data[1] - component 0 value
// data[2] - component 1 type
// etc
// so for 8 components it needs an array of 16 doubles
#define GETSET_SIMULATION_COMPONENT_VALUES 6

//
////////////////////////////////////////////////////////////////////////////////////////////
//
// Get or set a string parameter
// The functions need a parameter to say what is to be set/got - see details below,
// a string for the result / parameter
// and an index value
// OLE equivalents:
// int SetStringAutomation(LONG nWhat, LONG nIndex, BSTR newstring);
// BSTR GetStringAutomation(LONG nWhat, LONG nIndex);
__declspec(dllexport) int _stdcall MyVNASetString(int nWhat, int nIndex, _TCHAR * sWhat );
__declspec(dllexport) CString _stdcall MyVNAGetString(int nWhat, int nIndex );

//
// options for MyVNAGetString and MyVNASetString
// case 0 - equation
// nIndex is which one to set/get 0..3
#define GETSET_EQUATION 0
```

Annex E - Custom VNA Interface

The program supports a hardware interface called “Custom”. This provides an opportunity to implement a custom hardware interface.

Present in the installation directory of the program is a DLL named “CustomVNAinterface.dll”. This DLL may be replaced by another to interface to other hardware. The DLL must export an interface identical to the existing DLL. Available for download is a Visual Studio project named “CustomVNAinterface” which was used to build the existing DLL. The code currently present in the DLL implements a functional equivalent of the built-in USB N2PK VNA logic. If you wish to change the filename from “CustomVNAinterface.dll”, change the registry key “Custom VNA DLL name” located in the “Setup” key under “myVNA” using the registry editor.

By all means replace the code with your own but do not change anything in CustomVNAinterface.h other than the class CustomVNAhw which is used internally by the DLL and is not visible to the main program.

The current version is as shown below. myVNA will call CustomVNAhwInit() before calling any other function and will call CustomVNAhwFree() before closing down. For all called functions other than init, myVNA will pass to the called function the pointer returned by the init function. This provides a simple way for the custom VNA handler to manage its own internal class. For the definition of each function, see the sample code itself.

```
// CustomVNAinterfaceDLL.h

// this file includes definitions for the custom VNA interface DLL
// it must match the one used by myVNA so don't change definitions !

#pragma once

// do not change these definitions - they relate to the interpretation of return values by myVNA
// error flags used in USB and TCPIP comms
//
// ADC Read flags (also explicitly returned from CheckVNAReady and RescanForVNA)
#define VNAHW_NONETCONNECT (1<<10)
#define ADC_SAME (1<<9)
#define ADC_OVERRANGE (1<<8)
#define ADC_READ_TIMEOUT (1<<7)
#define VNA_NO_POWER (1<<6)
#define ADC_START_FLAG (1<<4)
#define ADC_NOT_RESPONDING (1<<3)
#define ADC_DATA_READY (1<<5)
// return codes from Read, Command and Reset functions
#define VNAHW_OK  0
#define VNAHW_HWERROR1
// the above are also returned as error codes from DoScan, along with the following
#define VNAHW_OVERRANGE 2
#define VNAHW_ADC_TIMEOUT 3
#define VNAHW_NO_POWER 4
// from MSA scan functions
#define VNAHW_INVALID_ADC 5
#define VNAHW_COMMSERROR 6

// These are the exported functions called by myVNA.
// Call CustomVNAhwInit() before anything else
// Call CustomVNAhwFree() before closing
// Pass the value returned by CustomVNAhwInit() as the first parameter to all other functions
// see the c++ source for details of each function

__declspec(dllexport) void *_stdcall CustomVNAhwInit(int nInstance);
__declspec(dllexport) void _stdcall CustomVNAhwFree(void *pHelper);
__declspec(dllexport) bool _stdcall CheckVNAHardware(void *pHelper);
```

```
__declspec(dllexport) bool _stdcall RescanForCustomVNA(void *pHelper, int nInstance);
__declspec(dllexport) bool _stdcall GetCustomVNACodeVersions(void *pHelper,
              int *pCustomVNAinterfaceDLLversion, int *pVnaDllVersion, int *pUSBCodeVersion );
__declspec(dllexport) int _stdcall CustomVNACommand(void *pHelper, double dClockDDS,
              double LoDDS, double RfDDS, int LoPhase, int RfPhase, int Delay,
              int AdcCount, int AdcMode, bool bPauseInOut, bool bOverlapDDSandADCFlag,
              bool bAdcOnly );
__declspec(dllexport) int _stdcall CustomVNARead(void *pHelper, int *pflags, int *pcount,
              signed long *pdata);
__declspec(dllexport) int _stdcall CheckCustomVNAReady(void *pHelper, int nInstance);
__declspec(dllexport) int _stdcall PowerDownCustomVNA(void *pHelper);
__declspec(dllexport) int _stdcall ResetCustomVNA(void *pHelper);
__declspec(dllexport) int _stdcall CustomVNASweep(void *pHelper, double dClockDDS,
              double dLoDDS, double dRfDDS, int LoPhase, int RfPhase,
              double dLoStepDDS, double dRfStepDDS, int LoStepPhase, int RfStepPhase,
              int nSteps, int nStepDelay, int nInitDelay, bool SetSW1,
              bool bPowerDownBetweenSweeps);
__declspec(dllexport) int _stdcall SetCustomVNASwitchLines(void *pHelper, int nSetting );
__declspec(dllexport) int _stdcall SetCustomVNAAttenuatorLines(void *pHelper, int nSetting );
__declspec(dllexport) int _stdcall SetCustomVNAPortABLines(void *pHelper, int nSettingA,
              int nSettingB );
__declspec(dllexport) int _stdcall CustomVNARawRead(void *pHelper, int *dataA, int *dataB);
__declspec(dllexport) int _stdcall CustomVNAGetInformation(void *pHelper, int StructVersion,
              int StructSize, int what, void *pResult);
```

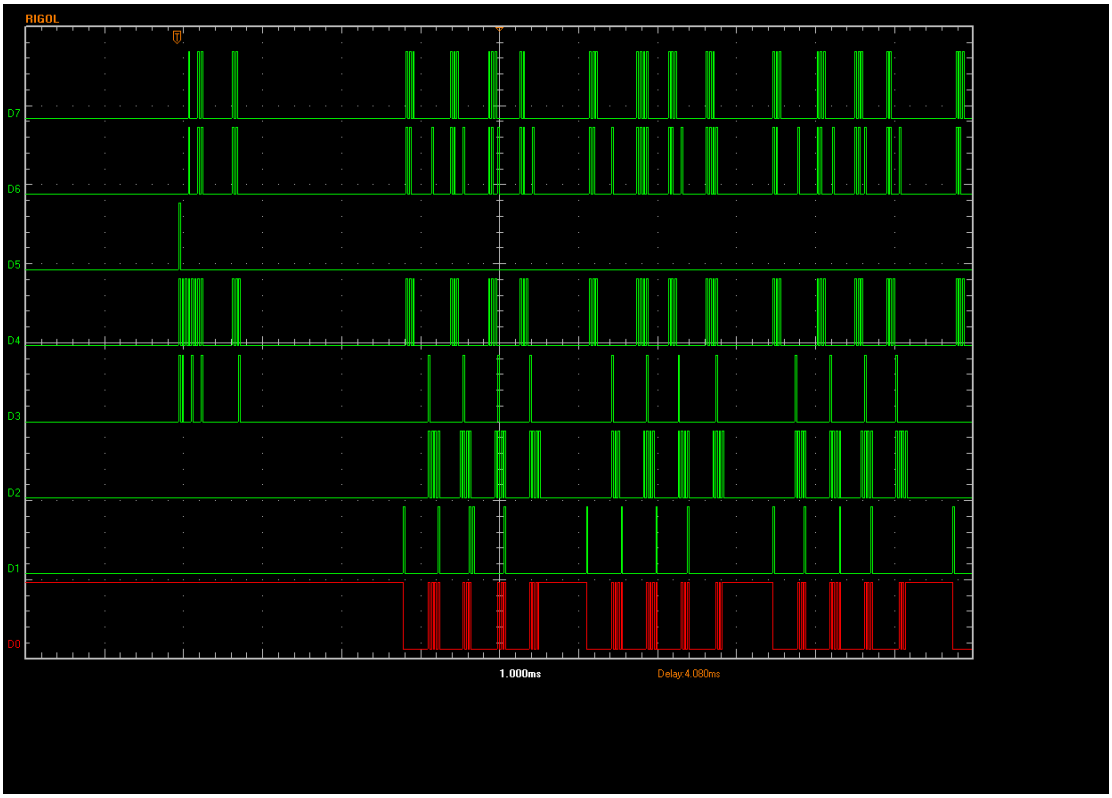
Annex F - How fast will it scan?

On the left of the progress bar in the status line is a trace speed estimation. Given the current sweep, it tries to predict the current sweep time. This takes into account the interface, steps, CDS mode, averages, single or dual detector, step delay, start delay and ADC mode. It does not try to tweak itself for the PC speed though – it is based on typical PC timings.

USB Timings

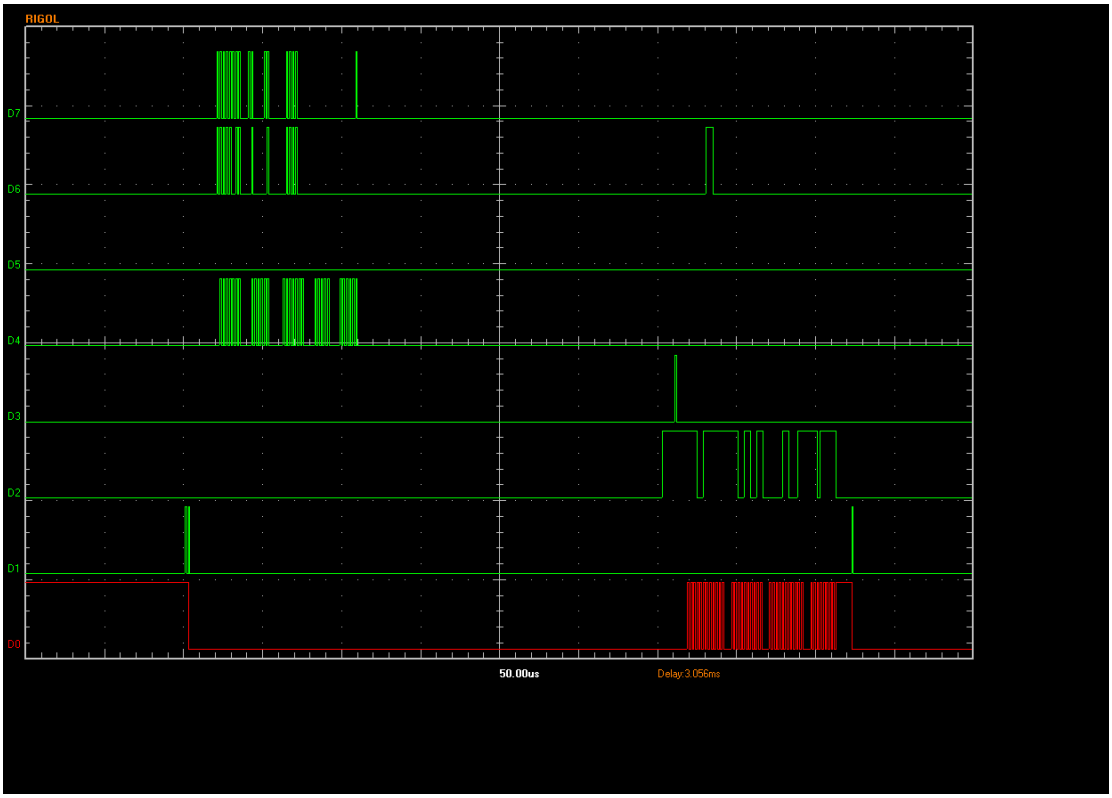
With USB, if you set all delays to zero and speed to 10, a 200 point CDS non dual detector scan will take 0.39 seconds. To the raw ADC conversion time (about 320 µsec) the program will add a fixed overhead of 110 µsec giving a per conversion time of about 430 µsec. If you select a step delay, this gets added to each step point, and there is a fixed scan startup time of 960 µsec which can be extended with the scan start delay setting. A rate of 430 µsec per conversion is a rate of just over 2300 conversions per second. It's not quite that simple though. USB frames are 125 µsec and the conversions will align with this grid in a manner that depends on delay settings. However, the data rate should be at least 2000 ADC conversions per second, including DDS update time. This data rate is sustained through the scan including screen update as long as the PC can keep up. An overall logic analyser trace of the first part of a scan is shown below. Note that in this scan the data lines are as follows

D7LO DDS data
D6RF DDS data
D5DDS Reset
D4DDS WCLK
D3DDS FQUD
D2ADC SDO
D1ADC CS
D0ADC SCLK



In the above trace the scan start delay is set to 2000 µs, the frequency step delay to 500 µs and the phase step delay to 0. ADC is fastest mode, 10, and DDS update is set to overlap ADC operation. On the left can be seen the reset of the DDS followed by a 2 division (200 µs) delay before the first ADC conversion. ADC conversions are then grouped in sets of 4 as this is a CDS scan with the programmed 500 µs delay between them.

Zooming in on a single ADC cycle yields the following trace. The ADC CS signal pulses and the clock goes low to start the ADC conversion. 20 µs later the DDS update occurs for the next point and the CPU then waits for the ADC conversion to finish. As soon as it detects this it pulses FQ_UD to change the DDS output and reads the ADCs. This is a dual detector scan, the timing is the same for single or dual detector. Immediately it has finished it starts the next ADC conversion.



Note that the DDS is being updated whilst an ADC conversion is taking place (ADC CS is low – D0) and DDS clock (D4) is active. This maximises speed. It is possible however that in low noise floor CDS scans at low ADC rates, this may cause digital noise in the conversion, so there is a setting under "VNA hardware" labelled "Load DDS during ADC" which if unticked, updates the DDS at the end of an ADC conversion as shown below. This slows the scan speed down adding 320 µs to an ADC conversion giving a maximum rate of 1560 ADC conversions per second.

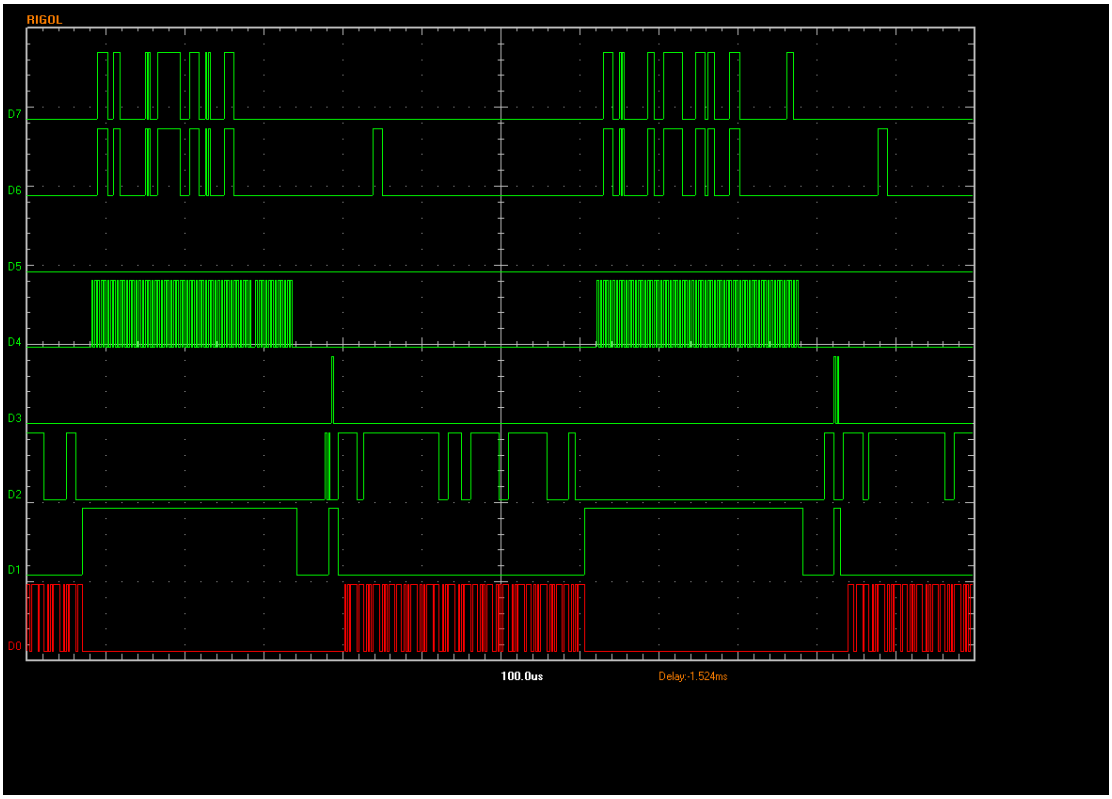
Note that when in continuous scan mode there is (on a typical PC[1]) a 4.5 ms delay between each sweep as the PS updates the display and starts a new scan.

Parallel Port

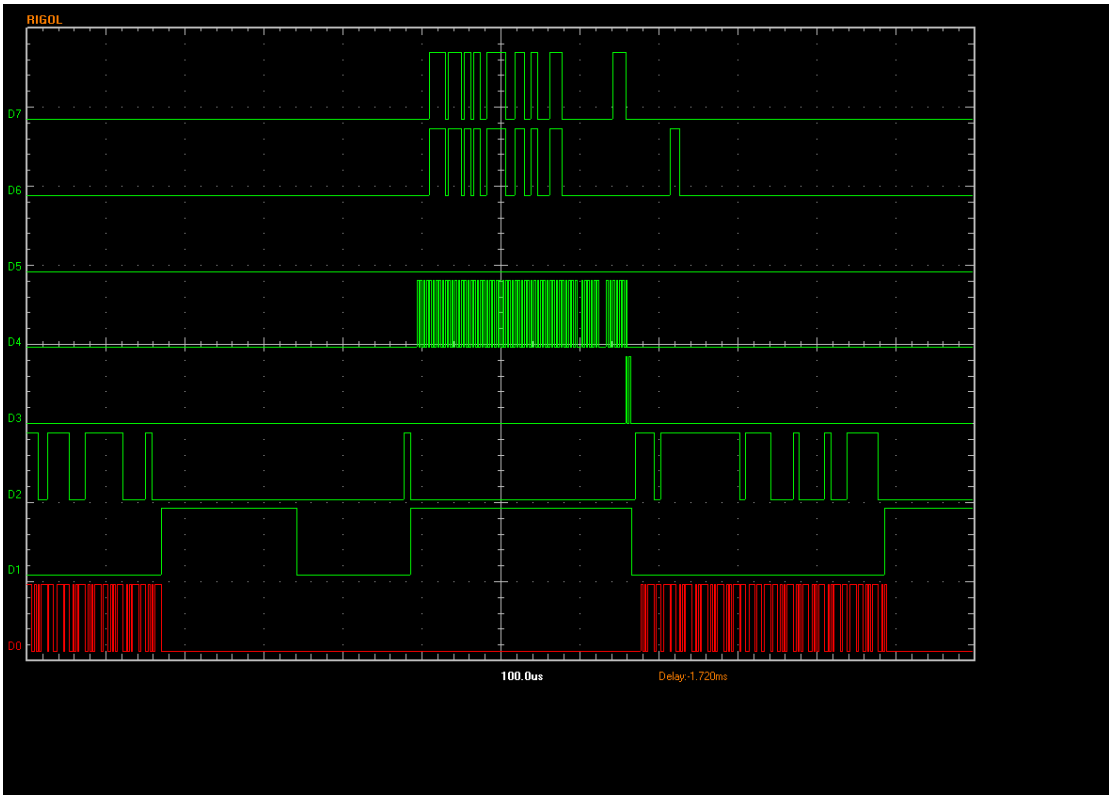
The parallel port is slower than USB primarily because inpout32.dll imposes a speed limit. WinIO may provide higher performance. Both sets of figures are given.

Here is a zero delay single ADC conversion using InpOut32 zooming in on a complete ADC cycle, indicating that the delay for ADC result reading is just over 300 µsec, giving an ADC cycle rate of just over 1600 points per second. With WinIO the ADC reading drops to 110 µsec giving over 2000 points per second. The following trace is InpOut32.

[1] which is an Athlon 64 X2 3800+ running 32 bit XP home edition for all the values given in this section of the manual



Finally, if you select not to overlap ADC conversion and DDS update the scan slows down some more adding another 260 µsec delay giving an ADC rate of 1100 conversions per second with InpOut32. With WinIO, the overall rate is 500 µs or about 2000 points per second on a typical PC.



Annex G - Configuring the USB hardware

This in an updated extract of the document “configuring USB adapter and installing drivers v2.0”, the full version of which may be found in the USB folder of the program installation directory.

Do not connect the VNA to the USB interface until you have successfully completed step 13!

1. Unzip the driver files into a suitable location unless (as is usually the case) they are already present in the USB drivers folder of the installation folder of myVNA. This will install the 'usb configure.exe' program and also create driver subdirectories called ‘XP’ and ‘Vista-Windows 7’. If installed as part of myVNA, a shortcut to the program will already be present under “myVNA” from the sart menu.

2. Plug in the USB interfce. Windows will identify there is a new USB device, and may then search for several minutes to find drivers; you have to wait until it gives up. (If you see any messages about "This device will go faster", ignore them and they will go away. See step 10 for more details.). Note: it is possible that, if previously unsigned drivers were loaded or Windows was instructed not to install the drivers, that no such response will be obtained. In this case, run Device Manager, locate the entry for the interface and select “update drivers” manually.

3. When the "Found New Hardware Wizard" eventually appears (and with slight variations depending on the operating system version):
   3.1 Choose "No, do not connect to Windows Update" and click Next.
   3.2 Choose "Install from a list or specific location" and click Next.
   3.3 Choose "Search for the best driver in these locations", check "Include this location" and Browse to one of the subdirectories created in step 1 as appropriate for the operating system. Click OK and then click Next.

4. The USB drivers are installed. Ignore any "EEPROM missing" error message and click “Next”. It will then tell you that the drivers have not passed Windows Logo testing. Ignore this and click "Continue Anyway" (all this means is that the chip has been found correctly and either the EEPROM is missing or it is blank). When the process completes, click the "Finish" button. You are returned to the Windows desktop and the "Found New hardware" flag pops up at the lower right. Wait for it to do this – driver installation is a two step process and both must complete correctly.

5. Go to the directory you created in step 1 and double-click the 'usb configure.exe' program. If you get a "Compressed Files warning", click "Run". The "N2PK USB Interface Control" dialog opens. Make sure that the title of the dialog box indicates version 2.2 or later of the program.

6. Go to the "Setup" tab and use the Browse" button in the "Vend_Ax filename" field to find the 'Vend_Ax.Hex' file. Click "Save Settings". Click on the “Find devices” button and select the device indicated. Note that if you have any other Cypress controller chips plugged in at this point then it may find these as well; be careful not to reprogram them inadvertently.

7. Go to the "Program FX2 Chip EEPROM" tab and click "Download".

Watch the messages at the bottom. If the download succeeds, the final line will say "SUCCESS: Program downloaded" and it will enable the "Read" and "Write" buttons on that tab. Click "Read" and you should get 8 bytes of FF. At this point you are communicating correctly with the USB card. If you navigate away from this tab in the program at this time then you will find you can no longer communicate with the card until you reset it by disconnecting it and reconnecting it.

8. In the "Write EEPROM" field of the same tab, select the "N2PK" option from the drop-down list and click "Write Data". That should result in a success message, and the EEPROM will have been programmed. Now click the "Read" button and check that it reads back the same string that was programmed in (see "Data to be written" at bottom of tab).

   If the EEPROM does not read back correctly, go to the drop-down list, select the "Erase (write FFs)" option and repeat the "Write Data" operation to (try to) reset the EEPROM. Do not pull the USB plug out while the EEPROM is not reading back correctly, or you will lose control of the EEPROM.

   When all is OK in step 8, close the 'usb configure.exe' program. Note that at this point the program running on the USB controller chip is the “Vend_Ax” programming utility and not the VNA program.

9. Pull the USB plug out and reconnect it. When the "Found New Hardware Wizard" appears:
   9.1 Choose "No, do not connect to Windows Update" and click Next.
   9.2 Choose "Install from a list or specific location" and click Next.
   9.3 Choose "Search for the best driver in these locations", check "Include this location" and Browse to the appropriate subdirectory created in step 1. Click OK and then click Next.

   If the wizard does not appear, open Control Panel, select the “System” Icon and from there select the “Hardware” tab. Then click the “Device Manager” button. Select the first item in the list and from the Action menu select “scan for hardware changes”. This should then cause Windows to find the device.

10. The N2PK drivers are installed. Ignore the "Has not passed Windows Logo Testing" message and click "Continue Anyway". When the process completes, click the "Finish" button. You are returned to the Windows desktop and the "Found New hardware" flag pops up.

    If the "Found New Hardware Wizard" appears again, repeat steps 9 and 10 until Windows can routinely find the USB interface when it is plugged in.

    Note: from now on, whenever you plug the USB interface into a USB1 port (as used in older PCs), you will see an irritating message that "This device can perform faster". Ignore it.

    Note 2: If the PC wants to reboot let it. The driver will in fact be working correctly but it is simpler to get it over with. You may also find that if you plug it in to a different USB port it wil reinstall the drivers again. This should only happen once per port though.

11. Re-run the 'usb configure.exe' program (see step 5). At the "Compressed Files warning", click "Run". The "N2PK USB Interface Control" dialog opens.

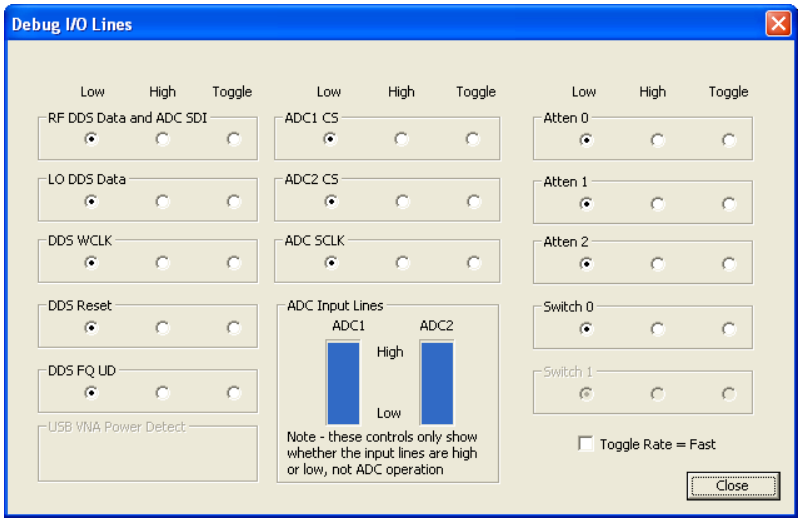
12. Go to the "Communicate with Adapter" tab and click "Check VNA interface". You should get an "N2PK VNA Interface found" message, and the "Read VNA Interface" and "Write VNA Interface" buttons will be enabled.

13. Click "Read VNA Interface" - still with no VNA connected - and you should read the message '00 00 X0 C0 00'. Where X is 8 or 4.

Annex H - Testing I/O Lines

There is a “test” function under “Instrument Modes” that allows low level debugging of I/O lines. It works for either USB or Parallel port mode.

When selected, it opens a dialog as shown



Each line may be set high, low or toggling. When toggling, the rate may be high (10 to 20 msec approx per change – depends on windows) or 0.5 second. There are two input lines (three for the USB interface). The two ADC Serial Data Out lines appear as two vertical progress bars representing high and low levels. In USB, there is also a power detect line. Its status is reflected in the bottom left of the dialog box.

Please note that the ADC input lines are not showing the result of an ADC conversion – they are just indicating binary high (all blue) or binary low (10% blue) according to whether the input line is high or low.

Note that the ADC lines are read at the same rate as the “Toggle” speed, so if set to slow, may take up to 1 second to update.

Index

Activation 8
Attenuator and Switch Control 81
Calibration 19
  Interpolation 54
  Introduction 16
  Isolation Averaging 20
  LLC Calibration 84
  Logarithmic Scans 71
  S Parameters 74
  Save to Disk 20
  Standard Offsets 21
  Summary Display 21
  Unguided Calibration 85
  Validity 20
Calibration Standards 83
  LLC 84
  Load & Save to File 83
  Open, Short, Load 83
Clipboard 51
  CSV and Excel 51
  Print Notes 51
  Report View 51
Crystal Motional Parameters .... See Equivalent Circuits
Cursors
  Dragging 39
  Edit Menu 40
  Horizontal 39
  Popup Menu 38
  Vertical 39
Custom VNA Interface 169
Display Options 17
  Autoscale 22, 25
  Equivalent Circuits 62
  Graticule 17
  Labelling Axes 19
  Print Notes Dialog 50
  Progress Bar 17
  Report View 60
  Smith/Polar View 56
  Snap to 125 26
  Status Bar 17
Display Sets 18
  Display Left 18
  Display Right 18
  Display Sets 18
Equivalent Circuits 62
  Crystal Transmission Jig 63
  Crystals 62
  K8ZOA 63
  Logging Results 65
  Refining Results 65
  Reflection 63
  Transmission 63
Full Screen Display 46
Group Delay 79
Hardware Settings 11
  ADC Mode 17
  Clock calibration 12
  DDS Power Down 130
  Interface Selection 14
  Interface Type 15
  MSA Hardware Setup 13
  Multiple USB VNAs 126
  N2PK Hardware Setup 12
  Parallel Drivers 15
  Reference Mode 89
  RFIV Configuration 88
  Vector Voltmeter 103
  VNA Hardware 11
Harmonic Modes 77
  Harmonic Suppression Mode 16
  Harmonics 77
Installation 7
  myVNA.msi 7
  Parallel Port 7
  Previously Installed Version 7
  Product Activation 8
  Product Key 8
  USB drivers 7
Instrument Mode 17
  Always do Dual Scan 68
  Dual Detector 68
  Logarithmic Scan 71
  Reference Mode 89
  Reflection 17
  Reflection Data Port 76
  Reverse Scans 73
  RFIV Mode 88
  S Parameters 73
  Signal Generator 101
  Test Hardware I/O 178
  Transmission 17
  Vector Voltmeter 103
Keyboard Shortcuts 46
Locking Axes 32
Logarithmic Displays 71
Marker Measurements 97
Markers 26, 42
  Global show/hide 46
  Interpolation 43
  Manual Markers 29
  Marker Arithmetic 28, 44
  Marker Display Location 29, 44
  Parameter 45
  Polar and Smith Chart 29, 44
  Q display 45
  Status Bar 29, 44
  Tracking 27
Mouse & Dragging 23
  Cursor Popup Menu 38
  Cursors 39

Double Click ... 32
Drag and Drop ... 52
Edit Cursors ... 40
Frequency Axis ... 24
Lock Scan and Display ... 38
Markers ... 29, 44
Moving the Display ... 24
Parts of the Display ... 37
Report View ... 60
Reset Label Positions ... 37
Vertical Scales ... 24
MSA ... 13
Configuration ... 13
Multiple Program Configurations ... 125
Network Client Server ... 128
Network Simulation ... *See* Simulation
OLE Automation ... 127
Popup Menus ... 31
Clear Scan Data ... 33
Copy to Clipboard ... 33
Decimal Places ... 32
Lock Axes ... 32
Log Scales ... 31
Manual Markers ... 33
Set Frequency ... 32
Set Scan to Current Frequencies ... 33
Vertical Axes ... 31
Printing ... 50
Header and Footer ... 50
Notes ... 50
Pen Width ... 50
Product Key ... 8
Program Limits and Options ... 53
Reference Mode ... 89
Reference Planes ... 87
Report View ... 60
Clipboard Copy ... 60
Dragging Columns ... 60
RF-IV ... 88
Configuration ... 88
S Parameters ... 73
10 & 12 term ... 73
5 & 6 term ... 73
Calibration ... 74
Reverse Scans ... 73
Test Sets ... 155
Saving and Loading Traces ... 47
Autosave ... 49
Extended Data Sets ... 47
File Formats ... 47
Saving to Disk ... 47
Touchstone ... 47
Scan Speed ... 171
Parallel Port ... 173
Status Bar Display ... 17
USB ... 171
Scanning ... 22
Locking Display and Scan ... 38
Single Scan ... 23
Signal Generator ... 101
Mode ... 101
Sweep ... 101
Simulation ... 114
Crystals ... 123
Example ... 118
Impedance ... 120
Limitations ... 124
Load & Save ... 120
Matching Networks ... 116
Recalculation ... 116
Setup ... 114
Step by Step Example ... 120
Synthesize Network ... 115
Smith and Polar Charts ... 56
Markers ... 56
Rescaling ... 57
Stored Traces ... 30
TDR mode ... 90
algorithm ... 91
bandpass mode ... 92
configuration ... 91
lowpass mode ... 92
Markers ... 91
mouse actions ... 91
Using Time Domain to tune variable coupling BPFs ... 93
Testing I/O Lines ... 178
ADC Lines ... 178
Toggle Rate ... 178
Trace Averaging ... 100
Trace Calculation ... *See* User Defined Display Functions
Network Simulation ... *See* Simulation
Show Marker Measurements ... 97
Transverters ... 80
Tutorial ... 10
ADC mode ... 17
Autoscale ... 23
Calibration ... 19
Clock calibration ... 12
Data stores ... 30
Display Options ... 17
Dragging things on the screen ... 23
First Run ... 10
First Scan ... 17
Hardware settings ... 11
Harmonic Mode ... 16
Interface selection ... 14
Markers ... 26
Snap to 125 ... 26
Toolbars ... 10
Treeview ... 10
User Defined Display Functions ... 108
Data Sources ... 109
Equation Editor ... 108
Functions ... 112
Numbers ... 113
Parameters ... 109
T Check ... 108
Values ... 113

Vector Voltmeter 103
Calibration 104
DDS Phase and Reset Test 106

End of Document